In this user manual we have tried to describe the matters concerning the operation of this CNC system to the greatest extent. However, it is impossible to give particular descriptions for all unnecessary or unallowable operations due to length limitation and products application conditions；Therefore, the items not presented herein should be regarded as “impossible” or “unallowable”.

Copyright is reserved to GSK CNC Equipment Co., Ltd. It is illegal for any organization or individual to publish or reprint this manual. GSK CNC Equipment Co., Ltd. reserves the right to ascertain their legal liability.

Preface

Your Excellency,

We are honored by your purchase of this GSK 988T Turning CNC System made by GSK CNC Equipment Co., Ltd.

This book is User Manual “Programming and Operation”.

To ensure safe and effective running, please read this manual carefully before installation and operation.

Warning

Accident may occur by improper connection and operation! This system can only be operated by authorized and qualified personnel.

Special caution:

The power supply fixed on/in the cabinet is exclusively used for the CNC system made by GSK.

It can't be applied to other purposes, or else it may cause serious danger!

Cautions

■ Delivery and storage

- Packing box over 6 layers in pile is unallowed.
- Never climb the packing box, stand on it or place heavy objects on it.
- Do not move or drag the products by the cables connected to it.
- Forbid collision or scratch to the panel and display screen.
- Avoid dampness, insolation and drenching.

■ Open-package inspection

- Confirm that the products are the required ones.
- Check whether the products are damaged in transit.
- Confirm that the parts in packing box are in accordance with the packing list.
- Contact us in time if any inconsistence, shortage or damage is found.

■ Connection

- Only qualified personnel can connect the system or check the connection.
- The system must be earthed, and the earth resistance must be less than 0.1Ω. The earth wire cannot be replaced by zero wire.
- The connection must be correct and firm to avoid any fault or unexpected consequence.
- Connect with surge diode in the specified direction to avoid damage to the system.
- Switch off power supply before plugging out or opening electric cabinet.

■ Troubleshooting

- Switch off power supply before troubleshooting or changing components.
- Check the fault when short circuit or overload occurs. Restart can only be done after troubleshooting.
- Frequent switching on/off of the power is forbidden, and the interval time should be at least 1 min.

ANNOUNCEMENT!

- **This manual describes various possibilities as much as possible. However, operations allowable or unallowable cannot be explained one by one due to so many possibilities that may involve with, so the contents that are not specially stated in this manual shall be regarded as unallowable.**

WARNING!

- **Please read this manual and a manual from machine tool builder carefully before installation, programming and operation, and strictly observe the requirements. Otherwise, products and machine may be damaged, workpiece be scrapped or the user be injured.**

CAUTION!

- **Functions, technical indexes (such as precision and speed) described in this user manual are only for this system. Actual function configuration and technical performance of a machine tool with this CNC system are determined by machine tool builder's design, so functions and technical indexes are subject to the user manual from machine tool builder.**
- **Though this system adopts standard operation panel, the functions of the keys on the panel are defined by PLC program (ladder diagram). It should be noted that the keys functions described herein are for the standard PLC program (ladder diagram).**
- **For functions and effects of keys on control panel, please refer to the user manual from machine tool builder.**

Safety Responsibility

Manufacturer's Responsibility

——Be responsible for the danger which should be eliminated and/or controlled on design and configuration of the provided CNC systems and accessories.

——Be responsible for the safety of the provided CNC systems and accessories.

——Be responsible for the provided information and advice for the users.

User's Responsibility

——Be trained with the safety operation of CNC system and familiar with the safety operation procedures.

——Be responsible for the dangers caused by adding, changing or altering to the original CNC systems and the accessories.

——Be responsible for the failure to observe the provisions for operation, adjustment, maintenance, installation and storage in the manual.

This manual is subject to change without further notice.

This manual is reserved by end user.

We are full of heartfelt gratitude to you for supporting us in the use of GSK's products.

Contents

Ⅰ PROGRAMMING 1

Chapter Ⅰ Programming Fundamentals 3

1.1 GSK988T Introduction 3
1.2 CNC system of machine tools and CNC machine tools 5
1.3 Programming Fundamentals 7
1.3.1 Coordinates definition 7
1.3.2 Increment system 9
1.3.3 Max. travel 10
1.3.4 Reference position 10
1.3.5 Machine coordinate system 10
1.3.6 Workpice coordinate system 11
1.3.7 Local coordinate system 11
1.3.8 Interpolation function 11
1.4 Coordinate Value and Dimension 12
1.4.1 Absolute programming and incremental programming 12
1.4.2 Diameter programming and radius programming 13
1.4.3 Decimal programming 14
1.4.4 Conversion between the metric and the inch 14
1.4.5 Linear axis and rotary axis 15
1.5 Structure of an NC Program 15
1.5.1 Program name 16
1.5.2 Block format 16
1.5.3 Word 17
1.5.4 Block number 26
1.5.5 Main program and subprogram 26
1.6 Program Run 27
1.6.1 Sequence of program run 27
1.6.2 Execution sequence of word 28

Chapter Ⅱ G Commands 29

2.1 Summary 29
2.1.1 G command classification 29
2.1.2 Omitting word input 31
2.1.3 Related definitions 33
2.2 Rapid Traverse (Positioning) G00 33
2.3 Linear Interpolation G01 34
2.4 Arc Interpolation G02, G03 35
2.5 Dwell G04 38
2.6 Cylindrical Interpolation 7.1 39
2.7 Polar Coordinate Interpolation G12.1, G13.1 43
2.8 Metric/Inch Switch G20, G21 45
2.9 Stored Travel Check G22, G23 45
2.10 Skip Interpolation G31 46
2.11 Automatic Tool Offset G36, G37 48
2.12 Reference Position Function 50

2.12.1 Reference position return G28 50
2.12.2 2nd, 3rd, 4th reference position return G30 51
2.13 Related Function of Coordinate System 52
2.13.1 Selecting machine coordinate system position G53 53
2.13.2 Workpiece coordinate system setting G50 54
2.13.3 Workpiece coordinate system selection command G54～G59 55
2.13.4 Local coordinate system setting G52 57
2.13.5 Level selection command G17～G19 59
2.13.6 Exact stop mode G61/cutting mode G64 59
2.14 Fixed Cycle Command 60
2.14.1 Axial cutting cycle G90 60
2.14.2 Radial cutting cycle G94 63
2.15 Multiple Cycle Commands 66
2.15.1 Axial Roughing Cycle G71 66
2.15.2 Radial Roughing Cycle G72 72
2.15.3 Closed Cutting Cycle G73 77
2.15.4 Finishing Cycle G70 82
2.15.5 Axial Grooving Multiple Cycle G74 83
2.15.6 Radial Grooving Multiple Cycle G75 86
2.15.7 Notes for multi cycle machining 89
2.16 Threading Cutting 90
2.16.1 Thread Cutting with Constant Lead G32 90
2.16.2 Thread cutting with variable lead G34 93
2.16.3 Thread cutting cycle G92 95
2.16.4 Multiple thread cutting cycle G76 97
2.17 Constant Surface Speed Control G96, Constant Rotational Speed Control G97 103
2.18 Feedrate per Minute G98, Feedrate per Rev G99 105
2.19 Drilling/Boring Fixed Cycle Command 106
2.19.1 End drilling cycle G83 /side drilling cycle G87 107
2.19.2 End Boring CycleG85 / Side Boring Cycle G89 111
2.19.3 Cancelling Drilling/Boring G80 112
2.19.4 Notes for Drilling/Boring Cycle 112
2.20 Tapping Cycle Command 112
2.20.1 Tapping Mode 113
2.20.2 End Rigid Tapping Cycle (G84) / Side Rigid Tapping Cycle (G88) 114
2.20.3 End Common Tapping Cycle (G84) /Side Common Tapping Cycle (G88) 120
2.21 Automatic Chamfering Function 123
2.22 Macro Command 126
2.22.1 Variable 126
2.22.2 System variable 127
2.22.3 Operation and jump command 131
2.22.4 Macro program statement and NC statement 136
2.22.5 Macro program call 136

Chapter Ⅲ MSTF Commands 139

3.1 M (Miscellaneous Function) 139
3.1.1 End of program M02 139

3.1.2 End of program run M30 139
3.1.3 Program stop M00 139
3.1.4 Optional stop M01 140
3.1.5 Subprogram call M98 140
3.1.6 Subprogram Call M198 141
3.1.7 Return from Subprogram M99 141
3.1.8 The Following M commands for standard ladder(some functions modified by K parameters) 142
3.1.9 M Commands defined by standard PLC ladder 143
3.2 Spindle Function 143
3.2.1 Spindle speed analog voltage control 143
3.2.2 Spindle override 144
3.3 Tool Function 144
3.3.1 Tool offset 144
3.3.2 Tool Life Management 147

Chapter IV Tool Nose Radius Compensation 151

4.1 Application 151
4.1.1 Overview 151
4.1.2 Imaginary tool nose direction 152
4.1.3 Compensation value setting 155
4.1.4 G40/G41/G42 command function 156
4.1.5 Compensation direction 157
4.1.6 Cautions 159
4.1.7 Application 160
4.2 Tool Nose Radius Compensation Offset Path 161
4.2.1 Inner and outer side 161
4.2.2 Tool traversing when starting tool 161
4.2.3 Tool traversing in Offset mode 163
4.2.4 Tool traversing in Offset canceling mode 168
4.2.5 Tool interference check 169
4.2.6 Commands for canceling compensation vector temporarily 171
4.2.7 Particulars 174

Ⅱ OPERATION 181

Chapter Ⅰ Overview 183

1.1 Operation Overview 183
1.2 System Setting 184
1.3 Display 185
1.4 System 187
1.4.1 System panel 187
1.4.2 System key definitions 188
1.5 Machine Operation Panel 190
1.5.1 Division of machine operation panel 190
1.5.2 State indicator and press key definition on the panel 191

Chapter Ⅱ Power on, Power off and Safety Protection 196

2.1 Power on 196
2.2 Power off 197
2.3 Overtravel Protection 197
2.4 Overtravel Protection in Memory Travel Limit 197
2.5 Emergence Operation 199
2.5.1 Reset 199
2.5.2 Emergency stop 199
2.5.3 Feed hold 199
2.5.4 Cutting off power supply 199

Chapter Ⅲ Windows 200

3.1 Position Display Window 205
3.1.1 Absolute coordinate window 206
3.1.2 Relative coordinate display 207
3.1.3 Machine coordinate display 208
3.1.4 Comprehensive coordinate 208
3.1.5 Setting the relative coordinate 209
3.1.6 Switching between the mode and the comprehensive message 210
3.1.7 Clearing workpiece count 211
3.1.8 Clearing run time 211
3.2 Program Window 212
3.2.1 Local directory and U disk directory 212
3.2.2 MDI program 213
3.2.3 Item/times 214
3.3 System Window 214
3.3.1 System parameter setting and rewriting window 215
3.3.2 Screw pitch compensation setting and rewriting window 218
3.3.3 System message and operation authority levels 219
3.3.4 System file management 222
3.3.5 Ladder diagram 223
3.4 Setting Window 229
3.4.1 Tool offset setting 229
3.4.2 CNC setting window 233
3.4.3 Macro variable window 238
3.5 Message Window 239
3.5.1 Alarm message check window 240
3.5.2 Alarm record check window 241
3.5.3 Diagnosis window 242
3.5.4 Oscillograph window 245
3.5.5 GSK-CAN window 248
3.6 Graph Window 249
3.6.1 Setting graph parameter 249
3.6.2 Processing graph path 250
3.6.3 Simulation graph 251
3.7 Help Windows 252

Chapter Ⅳ Editing and Managing a Program 254

4.1 Searching, Creating, Executing and Opening a Program 254
4.1.1 Searching a program 254
4.1.2 Creating a program 254
4.1.3 Executing a program 255
4.1.4 Opening a program 256
4.2 Renaming, Outputting, Deleting and Arraying Programs, Saving a Program as 257
4.2.1 Renaming a program 257
4.2.2 Saving a program as 258
4.2.3 Deleting a program 259
4.2.4 Outputting a program 259
4.2.5 Arraying programs 260
4.3 Editing and Rewriting a Program 260
4.3.1 Editing a program 260
4.3.2 Rewriting a program 261
4.3.3 Shortcut key 262
4.4 Block Comment 263
4.5 Generating a Block Number 263
4.6 Background Editing a Program 263

Chapter Ⅴ Manual Operation 264

5.1 Manual Reference Position Return 264
5.2 Manual Feed 265
5.3 Increment Feeding 266
5.4 MPG Feeding 267

Chapter Ⅵ Auto Operation 270

6.1 Auto Running 270
6.1.1 Selecting the running program 270
6.1.2 Program running 271
6.1.3 Running from any block 272
6.1.4 Skip 272
6.1.5 G31 skip 273
6.1.6 Stop auto running 273
6.2 MDI Running 274
6.2.1 Editing and running the program in MDI mode 274
6.2.2 Running from any block 275
6.2.3 Stop MDI running 275
6.3 DNC Running 275
6.4 Auto Running Control 278
6.4.1 Machine and miscellaneous function lock 278
6.4.2 Dry run 279
6.4.3 Single block running 280
6.4.4 Feedrate override 280
6.4.5 Rapid traverse override 281

Chapter Ⅶ Tool Offset and Setting Tools 282

7.1 Setting the Tool Offset and the Wearing Values 282

7.1.1 Direct input method 282
7.1.2 Measuring mode 283
7.1.3 +input mode 284
7.1.4 C input method 285
7.1.5 Clearing the offset value or the wearing value 286
7.2 Fixed-Point Tool Setting 287
7.3 Trial Cut Toolsetting 287
7.4 Position Record 290
7.5 Automatic Tool Compensation 290

Chapter Ⅷ Setting and Display Graphs 292

8.1 Setting the Graph Parameter 292
8.2 Path Graph Display and Operation 293
8.3 Simulation graph display and operation 294

Chapter Ⅸ U disk Use 296

9.1 Sending a Program 296
9.2 Backup Value 297
9.2.1 System file backup 297
9.2.2 Servo parameter backup 298

Chapter Ⅹ Processing Examples 301

10.1 Outer End Face Machining 301
10.2 Compound Machining 304

Chapter Ⅺ Parameters 310

11.1 Parameters Related to System Setting 311
11.2 Parameters Related to Interfaces of Input and Output 311
11.3 Parameters Related to Axis Control/Setting Unit 311
11.4 Parameters Related to Coordinate System 315
11.5 Parameters Related to the Stroke Detection 317
11.6 Parameters Related to Feedrate 320
11.7 Parameters Related to Control of Acceleration and Deceleration 324
11.8 Parameters Related to Servo and Backlash Compensation 326
11.9 Parameters Related to Input/Output 330
11.10 Parameters Related to Display and Editing 332
11.11 Parameters Related to Programming 334
11.12 Parameters Related to Screw Pitch Error Compensation 336
11.13 Parameters Related to the Spindle Control 339
11.14 Parameters Related to the Tool Compensation 344
11.15 Parameters Related to the Canned Cycle 347
11.15.1 Parameter of the Drilling Canned Cycle 347
11.15.2 Parameters Related to the Thread Cutting Cycle 348
11.15.3 Parameters Related to the Combined Canned Cycle 348
11.16 Parameters Related to the Rigid Tapping 349
11.17 Parameters Related to the Polar Coordinate Interpolation 351
11.18 Parameters Related to the User Macro Program 352

11.19 Parameters Related to Skip Function....353
11.20 Parameters Related to Graphic Display....355
11.21 Parameters Related to Run Hour and Parts Count Display....355
11.22 Parameters Related to MPG Feed....356
11.23 Parameters Related to PLC Axis Control....357
11.24 Parameters Related to Basic Function....360
11.25 Parameters Related to GSK-CAN Communication Function....361

Appendix 1 Alarm List....363

1.1 Program Alarms (P/S Alarms)....363
1.2 Parameter Alarms....372
1.3 Pulse Encoder Alarms....373
1.4 Servo Alarms....373
1.5 Overtravel Alarms....373
1.6 Spindle Alarms....374
1.7 System Alarms....374
1.8 Communication prompt on the operation panel....375
1.9 GSK-CAN Communication Prompts....375
1.10 Servo Inner Alarms....377

Appendix 2 Standard Ladder Function Allocation....381

2.1 X, Y Addresses Definition....381
2.2 Standard Operation Panel....384
2.2.1 Address X....384
2.2.2 Address Y....386
2.3 Standard PLC Parameter Instruction....389
2.3.1 Parameter K....389
2.3.2 Parameter DT....390
2.3.3 Parameter DC....391
2.3.4 Parameter D....391
2.4 PLC(Address A) Alarms (the Followings are Referred to V2.03b)....392

Appendix 3 Installation....394

3.1 GSK988T Appearance Dimension....394
3.2 Machine Operation Panel MPU02A of GSK988T....395
3.3 Machine Operation Panel MPU02B Appearance dimension of GSK988T....396
3.4 GSK988T-H Appearance Dimension....397
3.5 Appearance Dimension of GSK988T-H Operation panel....397

Appendix 4 Operation List....399

Ⅰ PROGRAMMING

Chapter I Programming Fundamentals

1.1 GSK988T Introduction

GSK988T is exclusive to the slant bed CNC turning machine and turning center with the horizontal and the vertical structures. It uses 400MHz high-performance process to control 5 feed axes(including Cs axis) and 2 spindles, communicates with the servo unit through GSK-CAN serial bus, and its matched servo motor uses the high-resolution absolute encoder to realize 0.1μm position precision, which can meet the requirements of high-precision turning and milling compound machining. It has the network interface to support the remote monitor and file transmission and to meet the network teaching and workshop management. GSK988T is the best choice for the slant bed CNC turning and turning center.

Fig. 1-1 GSK988T appearance

Technical characteristics

5 feed axes（including Cs axis）, 3-axis link, 2 analog spindles to realize the turning, milling compound machining

Command unit 1μm and 0.1μm, max. speed 60m/min（max. speed 24m/min in 0.1μm）

Optional to GSK-CAN servo unit to read/write the servo parameter and monitor servo unit

Extended I/O unit and GSK-CAN axis through serial bus

Nested many PLC programs, on-line editing, real-time monitoring PLC ladder

Part programs edited on the background

Network interface, remote monitoring and file transmission

USB interface, U disc file operation, system allocation and software upgrading

8.4 inch truecolor LCD, two-dimensional motion path and solid graph display

Technical specifications

Controllable axes

Max. controllable axes：5（including Cs axis）
Max. link axes：3

PLC controllable axes：5

Feed axis function

Least command unit：0.001mm, 0.0001mm
Least command range：±99999999× least command unit

Rapid traverse speed：max. 60m/min in 0.001mm command unit, max. 24m/min in 0.0001mm command unit

Rapid override：F0, 25%, 50%, 100% real-timing tuning

Cutting feedrate:

0.01 mm/min～60000 mm/min or 0.01 inch/min～4000 inch/min（G98: feed per minute）

0.01 mm/rev～500 mm/r or 0.01 inch/rev～9.99 inch/rev（G99: feed per revolution）

Feedrate override：0～150% 16-level real-time tuning

Interpolation mode: linear, arc, thread, polar interpolation, and rigid tapping

Thread function

Thread type: constant pitch straight thread/taper thread/end thread, variable pitch straight thread/taper thread/end thread

Thread head：1~99 heads

Thread pitch： 0.01mm～500mm（metric thread）or 0.01inch～9.99inch（inch thread）

Thread run-out：thread lenght, angle, speed can be set

Acceleration/deceleration function

Cutting feed: linear, exponential
Rapid traverse: linear

Thread cutting: linear, exponential

Initial speed, terminal speed and time of acceleration/deceleration are set by the parameter

Spindle function

2-channel 0V～10V analog voltage output，2-channel spindle encode feedback, double-spindle control

Spindle speed: spindle speed specified by S or PLC signal, its range: 0rpm～20000rpm

Spindle override：50%～120% 8-level real-time tuning

Spindle constant surface control

Rigid tapping

Tool function

Tool length compensation（tool offset）：99 groups
Tool wear compensation：99 groups of tool wear compensation data

Tool nose radius compensation（C type）

Toolsetting mode: fixed-point toolsetting, trial-cutting toolsetting, reference position return toolsetting

Offset execution mode: modifying coordinate mode, tool traverse mode

Precision compensation

Backlash compensation: compensation range (-9999~9999)× check unit
Memory pitch error compensation：1024 compensation points，compensation point number of each is set by the parameter, each point compensation range (-700~700) × check unit

PLC function

13 basic commands, 30 functional commands

PLC ladder on-line edit, real-time monitoring

2-level PLC program, up to 5000 steps, the 1st level program refresh period

Many PLC programs（up to 16 programs），the current running PLC program can be selected

I/O unit

Basic I/O：40 input /32 output

Operation panel I/O：96 input/96 output

Human-computer interface

Display in Chinese, English and others

Two-dimensional tool path and solid graph display

Servo state monitoring

Servo parameter on-line allocation

Real-time clock

On-line help

Operation management

Operation mode: Auto, Manual, Edit, MDI, DNC, MPG, Reference position return

Multi-level operation Authorization Management

Alarm log

Timed stop

Program edit

Program capacity：36M, 10000 programs（including subprogram and macro program）

Edit mode: full-screen edit, part program edit on the background

Edit function：searching, modifying and deleting program/block/word, copying/deleting block

Program format: ISO code, word without blank space, relative coordinates, absolute coordinate compound programming

Macro command: statement macro command program

Program call: macro program call with parameters, 12-level subprogram nesting

Grammar check: executing the rapid grammar check for the program(do not run the program) after it has been edit

Communication function

RS232 interface: part program and parameter transmission, DNC machining, upgrading PLC program and system software U disc

USB：U disc file operation, U disc file directly machining, upgrading PLC program and system software U disc

LAN：remote monitoring, network DNC machining, file transmission, remotely upgrading PLC program, system software

Safety function

Emergency stop

Hardware travel limit

Many storage travel checks

Data backup and recover

1.2 CNC system of machine tools and CNC machine tools

CNC machine tool is an electro-mechanical integrated product, composed of Numerical Control Systems of Machine Tools, machines, electric control components, hydraulic components, pneumatic components, lubricating, cooling and other subsystems (components), and CNC systems of machine tools are control cores of CNC machine tools. CNC systems of machine tools are made up of computerized numerical control(CNC), servo (stepper) motor drive devices, servo (or stepper) motor

etc.

Operational principles of CNC machine tools: according to requirements of machining technology, edit user programs and input them to CNC, then CNC outputs motion control commands to the servo (stepper) motor drive devices, and last the servo (or stepper) motor completes the cutting feed of machine tool by mechanical driving device; logic control commands in user programs to control spindle start/stop, tool selections, cooling ON/OFF, lubricant ON/OFF are output to electric control systems of machine tools from CNC, and then the electric control systems control output components including buttons, switches, indicators, relays, contactors and so on. Presently, the electric control systems are employed with Programmable Logic Controller (PLC) with characteristics of compact, convenience and high reliance. Thereof, the motion control systems and logic control systems are the main of CNC machine tools.

The system has simultaneously motion control and logic control function to control two axes of CNC machine tool to move, and has PLC function. Edit PLC programs (ladder diagram) according to requirements of input and output control of machine tool and then download them to GSK988T Turning Machine CNC system, which realizes the required electric control requirements of machine tool, is convenient to electric design of machine tool and reduces cost of CNC machine tool.

Softwares used for controlling GSK988T Turning Machine CNC system are divided into system software (NC for short) and PLC software (PLC for short). NC system is used for controlling display, communication, edit, decoding, interpolation and acceleration/deceleration, and PLC system for controlling explanations, executions, inputs and outputs of ladder diagrams.

Standard PLC programs are loaded (except for the special order) when GSK980TDa Turning Machine CNC System is delivered, concerned PLC control functions in following functions and operations are described according to control logics of standard PLC programs, marking with “Standard PLC functions” in GSK980TDa Turning CNC System User Manual. Refer to Operation Manual of machine manufacturer about functions and operations of PLC control because the machine manufacturer may modify or edit PLC programs again.

Programming is a course of workpiece contours, machining technologies, technology parameters and tool parameters being edit into part programs according to special CNC programming G codes. CNC machining is a course of CNC controlling a machine tool to complete machining of workpiece according requirements of part programs. Technical flow of CNC machining is shown in Fig. 1-2.

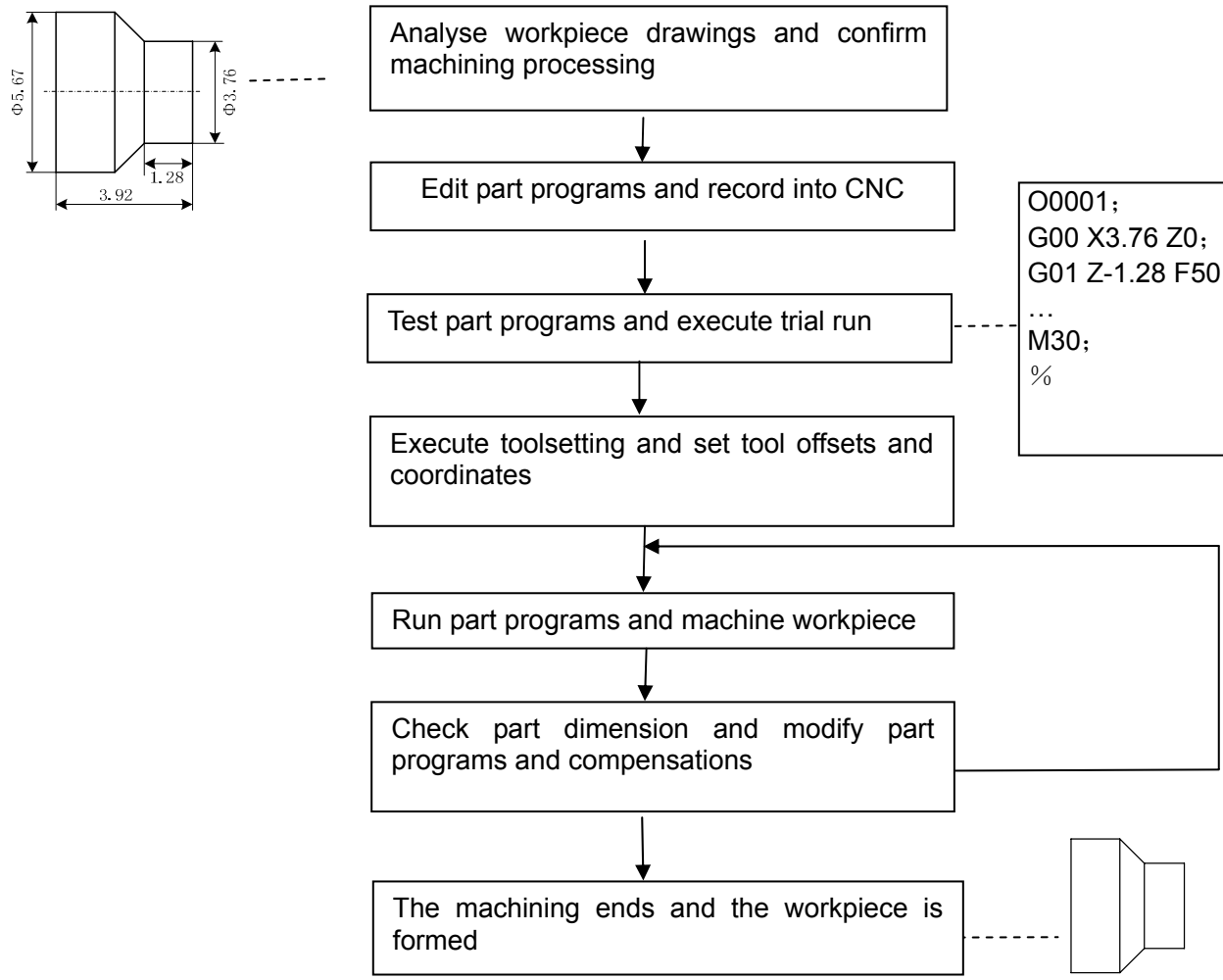


Fig. 1-2

1.3 Programming Fundamentals

1.3.1 Coordinates definition

The following figure is the sketch of CNC turning:

Fig. 1-3

GSK988T uses a rectangular coordinate system composed of X, Z axis. X axis is perpendicular with axes of spindle and Z axis is parallel with axes of spindle; negative directions of them approach to the workpiece and positive ones are away from it.

Parameter NO.1020 can set and modify program names for each axis and their responding relationship is as follows:

Table 1-3（a）

Axis name	Setting value	Axis name	Setting value
X	88	Z	90
Y	89	A	65
B	66	C	67

There is a front tool post and a rear tool post of NC turning machine according to their relative position between the tool post and the spindle, Fig. 1-5 is a coordinate system of the front tool post and Fig. 1-6 is a rear toolpost one. It shows exactly the opposite of X axes, but the same of Z axes from figures. In the manual, it will introduce programming application with the front tool post coordinate system in the following figures and examples.

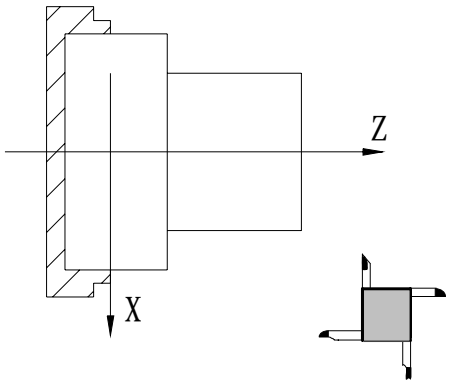


Fig.1-4 Front tool post coordinate system

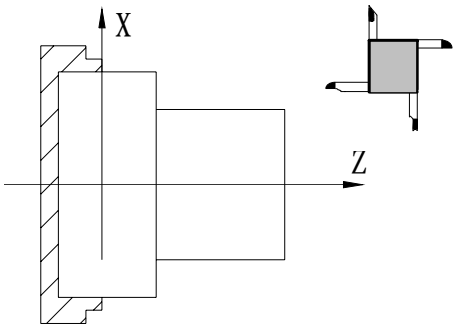


Fig.1-5 Rear tool post coordinate system

1.3.2 Increment system

Increment system includes least input increment (input) and least command increment (output). Least input increment is the least unit of programming movement distance. Least command increment is the least unit of tool movement on the machine tool. Their unit: mm, inch or degree. Increment systems are separately IS-B and IS-C. Bit 1 of NO. 1004 decides to select IS-B or IS-C. Bit 1 (ISC) setting of No.1001 is applied to all axes. For example: increment system of all axes is set to IS-C when the parameter selects IS-C.

Table 1-3（b） increment system IS-B

		Least input increment	Least command increment
Metric machine	mm input	0.001mm（diameter） 0.001mm（radius） 0.001deg	0.0005mm 0.001mm 0.001deg
	Inch input	0.0001inch（diameter） 0.0001inch（radius） 0.001deg	0.0005inch 0.001inch 0.001deg
Inch machine	mm input	0.001mm（diameter） 0.001mm（radius） 0.001deg	0.00005mm 0.0001mm 0.001deg
	Inch input	0.0001inch（diameter） 0.0001inch（radius） 0.001deg	0.00005inch 0.0001inch 0.001deg

Table 1-3（c） increment system IS-C

		Least input increment	Least command increment
Metric machine	mm input	0.0001mm（diameter） 0.0001mm（radius） 0.0001deg	0.00005mm 0.0001mm 0.0001deg
	Inch input	0.00001inch（diameter） 0.00001inch（radius） 0.0001deg	0.00005inch 0.0001inch 0.0001deg
Inch machine	mm input	0.0001mm（diameter） 0.0001mm（radius） 0.0001deg	0.000005mm 0.00001mm 0.0001deg
	Inch input	0.00001inch（diameter） 0.00001inch（radius） 0.0001deg	0.000005inch 0.00001inch 0.0001deg

Whether the least input increment is mm or inch is determined by the machine based on the parameter INM(1001#0). The least input increment can be switched between the inch and the mm input, which is controlled by G codes(G20 or G21) or the set parameter.

1.3.3 Max. travel

Max. travel=least command increment X（±）99999999

Table 1-3 (d) max. travel IS-C

Increment system		Max. travel
IS-B	Metric machine system	±99999.999mm ±99999.999deg
	Inch machine system	±9999.9999inch ±9999.9999deg
IS-C	Metric machine system	±9999.9999mm ±9999.9999deg
	Inch machine system	±999.99999inch ±9999.9999deg

Note 1: The unit is diameter value in diameter programming, is radius value in radius programming in the above table.

Note 2: The input command cannot exceed max. travel command.

Note 3: The actual travel decides the machine tool.

1.3.4 Reference position

Reference position is a fixed point on the machine tool. The tool can move to the position by executing the reference position return function. Generally, the reference position is used to tool change and setting coordinate system. GSK988T Turning CNC System can set 4 reference positions by parameters as follows:

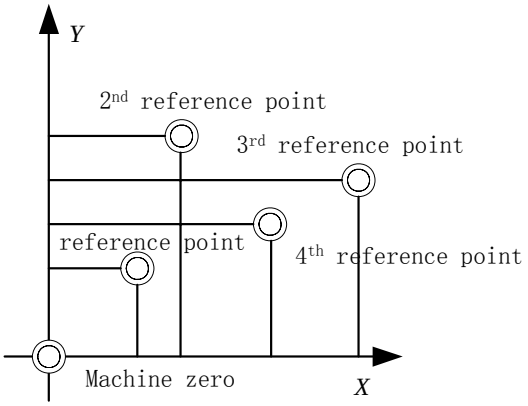


Fig. 1-6 reference position

1.3.5 Machine coordinate system

Machine tool coordinate system is a benchmark one used for CNC counting coordinates and a fixed one on the machine tool. **Machine tool zero** is a fixed point which position is specified by zero switch or zero return switch on the machine tool. Usually, the zero return switch is installed on max. stroke in axis positive direction. After the system is turned on, the reference position return is executed to set machine coordinate system. The machine coordinate system is not keeping until the system is turned off.

Note: For the machine with the incremental encoder, must execute the reference position return every time to set the machine coordinate system after power-off; for the machine with the multi-coil absolute encoder, need not execute the reference position return every time after power-off.

1.3.6 Workpice coordinate system

The workpiece coordinate system is a rectangular coordinate system based on the part drawing, also called floating coordinate system. The workpiece coordinate system is set by the system in advance, can be changed by moving its coordinate origin point. The established workpiece is valid till it is replaced by a new one. The system has preset 6 workpice coordinate systems (G54-G59).

1.3.7 Local coordinate system

When the system compiling programs in the workpiece coordinate system, sub-coordinate system of workpiece coordinate system can be set for easily programming, called local coordinate system as follows:

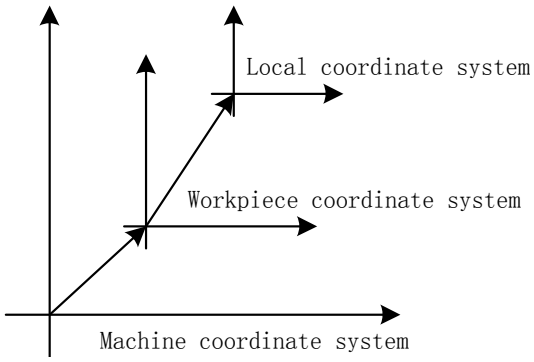


Fig. 1-7 local coordinate system

1.3.8 Interpolation function

Interpolation is defined as a planar or three dimensional contour formed by path of 2 or multiple axes moving at the same time, also called **Contour control**. The controlled moving axis is called link axis when the interpolation is executed. The moving distance, direction and speed of it are controlled synchronously in the course of running to form the required Composite motion path. Positioning control is defined that motion end point of one axis or multiple axes instead of the motion path in the course of running is controlled.

GSK988T has linear, arc and thread interpolation function.

Linear interpolation: Composite motion path of X, Z axis is a straight line from starting point to end point.

Circular interpolation: Composite motion path of X, Z axis is arc radius defined by R or the circle center (I, K) from starting point to end point.

Thread interpolation: Moving distance of X or Z axis or X and Z axis is defined by rotation angle of spindle to form spiral cutting path on the workpiece surface to realize the thread cutting. For thread interpolation, the feed axis rotates along with the spindle, the long axis moves one pitch when the spindle rotates one rev, and the short axis and the long axis directly interpolate.

Note 1:Xp, Yp, Zp are separately X or its parallel axis, Y or its parallel axis, Z or its parallel axis. The followings are the same as those.

Note 2: IP expresses the combination of X_Y_Z_(used in programming).

Example:

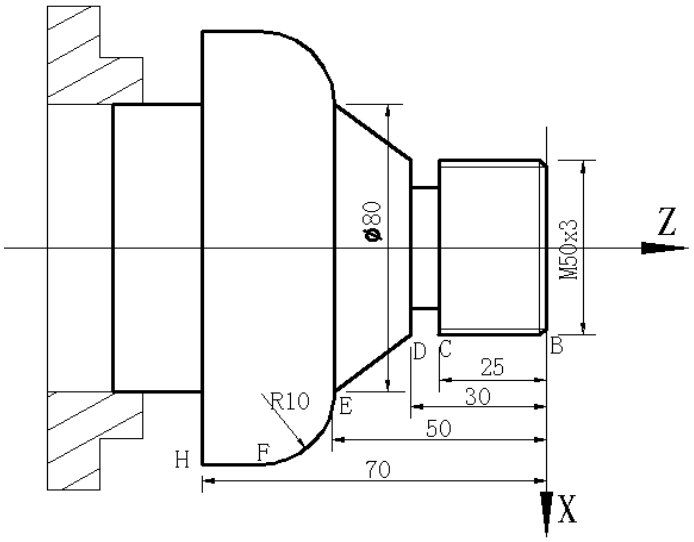


Fig.1-8

```
…
G32 W-27 F3;          （B→C; thread interpolation）
G1 X50 Z-30 F100;
G1 X80 Z-50;          （D→E; linear interpolation）
G3 X100 W-10 R10;     （E→F; arc interpolation）
…
M30;
```

1.4 Coordinate Value and Dimension

1.4.1 Absolute programming and incremental programming

The system has two methods to command the too traverse: absolute value and incremental value command. In the absolute programming, use the coordinate value programming of the end point; in the incremental programming, use the traverse distance programming. In the system, using the absolute programming or incremental programming is depended on the word of the command as follows:

Table 1- 4（a）

	Absolute value command	Incremental value command
X movement command	X	U
Y movement command	Y	V
Z movement command	Z	W
C movement command	C	H
A movement command	A	None
B movement command	B	None

The system can select the incremental programming or the absolute programming mode, or the incremental/absolute compound programming; the absolute command and the incremental command can be in the same block as follow:

X100.0 W100.0;

When the absolute command and the incremental command of one axis are in the same block, the following command value is valid.

The axis word can exist repetitively in the same block and the later value is valid, but when No.3403 Bit 6 (AD2) is set 1, the alarm occurs. U, W in other G command has bee specified to others. For example: in G73, the above conditions

1.4.2 Diameter programming and radius programming

Because the workpiece section is the circle in CNC turning controlled program, X dimension can use two kind of method; diameter programming command and radius programming command.

1. The user can select the radius programming or diameter programming, which is set by state parameter (No. 1006 Bit 3(DIAX)).
2. Parameters related to diameter/radius programming:

State parameter No.1006 BIT3 (DIAx):

0—radius programming；

1—diameter programming；

State parameter No.5004 Bit1(ORC):

0—offset value is expressed with diameter;

1—offset value is expressed with radius;

Pay more attention to the conditions in the following table when X uses diameter programming:

Table 1- 4 (b) related addresses and data to the diameter or radius programming

	Word	Explanation	Diameter programming	Radius programming
Related addresses to diameter/radius programming	**X**	X coordinate, polar coordinate	Diameter value	Radius value
		G50 sets X coordinate	Diameter value	Radius value
	U	X increment	Diameter value	Radius value
		G71 infeed amount	Radius value	
		X finishing allowance in G71, G72, G73	Parameter definition	
		tool retraction amount in G73	Radius value	
	R	Clearance in G71, G72	Radius value	
		Clearance after cutting in G75	Diameter value	Radius value
		Clearance to end point in G74	Diameter value	Radius value
		Taper in G90, G92, G94, G76, radius in G02, G03, thread finishing amount in G76	Radius value	
	I	X amount of circle center	Radius value	
	F	G32,G34,G92,Pitch long axis is X in G76	Radius value	
		X feedrate display	Radius/rev, radius /min	
Others	X or U value of position window	Display	Diameter value	Radius value

Note: Besides the above-mentioned addresses and data related to the diameter programming or the radius programming, other related to word and data related to X numerical value are expressed with radius value.

1.4.3 Decimal programming

Value can be input by decimal programming. Distance, time and speed can be input by decimal programming. The following addresses can use decimal point: X, Y, Z, A, B, C, U, V, W, H, I, J, K, R and F, and other addresses cannot use decimal programming.

There are two types of decimal point usage which is decided by No. 3401 Bit0(DPI).

When NO.3401 Bit 0(DPI) is set to 1, value without decimal point is with mm, inch.

When NO.3401 Bit0(DPI) is set to 0, input value is specified by least input increment.

Parameter setting			Least command unit
ROTx=0 Rotary axis	Rotary axis is not related to parameter INI	ISC=0（ISC system）	0.001deg
		ISC=1（ISB system）	0.0001deg
ROTx=1 Linear axis	INI=0 Metric	ISC=0（ISC system）	0.001mm
		ISC=1（ISB system）	0.0001mm
	INI=1 Inch	ISC=0（ISC system）	0.0001inch
		ISC=1（ISB system）	0.00001inch

Example: when the metric input, the least input increment unit are set to 0.001:

Program command	The corresponding actual value when DPI is 1	The corresponding actual value when DPI is 0
X1000 without decimal command value	1000mm Unit：mm	1 mm Unit: least input increment(set to 0.001)
X1000.0 with decimal command value	1000mm unit：mm	1000mm Unit：mm

The decimal which is less than the least input increment unit is discarded in course of program being executed.

Example: X2.34567. When the least unit of input increment is 0.001mm, X2.34567 becomes X2.345, when the least unit is 0.0001inch, it becomes X2.3456.

The system alarms when the specified is more than 8-digit value.

1.4.4 Conversion between the metric and the inch

Metric input or inch input is set by NO.0000 Bit2(INI). G commands corresponding to metric/inch system is as follows:

G20： inch input ；

G21： mm input.

Input data unit becomes the inch or metric input unit when NO.0000 Bit2 (INI) setting is changed. But, the angle unit is not changed.

Input data unit becomes the inch or metric input unit when NO.0000 Bit2 (INI) setting is changed. But, the angle unit is not changed.

——F feedrate;

——position command;

——zero offset of workpiece;

——tool compensation value;

——graduation unit of MPG;

——movement distance in incremental feed.

NO.1001 Bit0 (INM) can set MM or INCH input of least command increment in linear axis.

0：mm input(metric machine)

1：inch input（inch machine）

1.4.5 Linear axis and rotary axis

NO.1006 Bit0(ROTx) can set each axis to linear axis or rotary axis. NO. 1006 Bit 1 (ROSx) can be used to select the rotary type for each axis.

Absolute coordinate value is displayed circularly with the movement per rev set by NO.1260 when the cycle function is executed, which can prevent the rotary axis from overflowing. The cycle function is valid when NO.1008 Bit 0(ROAx) is set to 1.

For absolute value command, the coordinate values is the corresponding angle cycle value of per rev set by NO. 1260 after the machine moves. When NO.1008 Bit 1(RABx) is set to 0, the machine rotates according to the shortest distance(to the target point). For incremental command, the machine moves according to the angle defined by the command.

1.5 Structure of an NC Program

User needs to compile part programs (called program) according to command formats of CNC system. CNC system executes programs to control the machine tool movement, the spindle starting/stopping, the cooling and the lubricant ON/OFF to complete the machine of workpiece.

Program example:

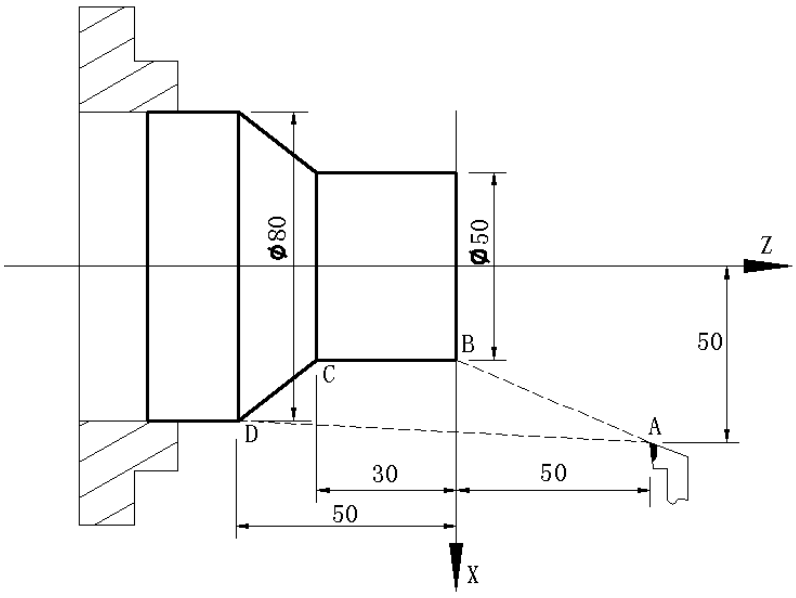


Fig. 1-9

O0001	;	(Program name)
N0005	G0 X100 Z50;	(Rapidly positioning to A point)
N0010	M12;	(Clamping workpiece)
N0015	T0101;	(Changing No.1 tool and executing its offset)
N0020	M3 S600;	(Starting the spindle with 600 r/min)
N0025	M8	(Cooling ON)
N0030	G1 X50 Z0 F600;	(Approaching B point with 600mm/min)

N0040	W-30 F200;	(Cutting from B point to C point)
N0050	X80 W-20 F150;	(Cutting from C point to D point)
N0060	G0 X100 Z50;	(Rapidly retracting to A point)
N0070	T0100;	(Canceling the tool offset)
N0080	M5 S0;	(Stopping the spindle)
N0090	M9;	(Cooling OFF)
N0100	M13;	(Releasing workpiece)
N0110	M30;	(End of program, spindle stopping and Cooling OFF)

The tool leaves the path of A→B→C→D→A after the above-mentioned programs are executed.

A program consists of a sequence of blocks, beginning with "OXXXX"(program name)and ending with "%"; a block begins with block number (omitted) and ends with ";" or "*". See the general structure of program as Fig. 1-10:

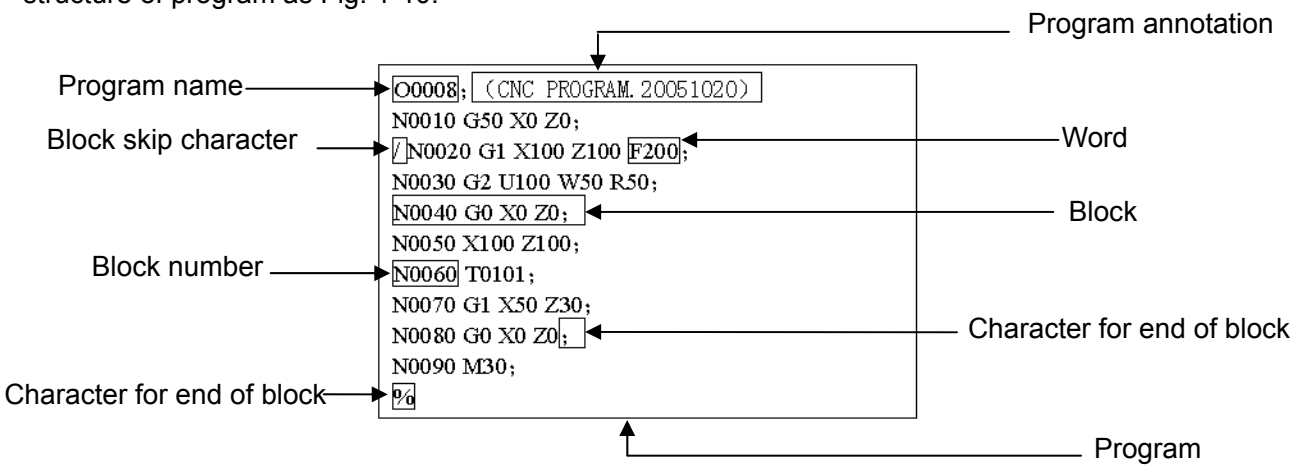


Fig. 1-10 Structure of a program

1.5.1 Program name

Format: ○ △△△△

Program number (0000～9999, the leading zero can be omitted）

Address O

△△△△ is number of a program name, its range is 4-digit integer 0000～9999, the system alarms when the negative program name is input. The system ignores NC commands when program are edited and other NC commands are edited in the first line.

1.5.2 Block format

1. Format: / N△△△△ countless words;

/: skip character. A block can have or not it, generally, it is placed in the initial position of a program; user can press "SKIP" on the operation panel to execute the operation when the skip function is valid, otherwise, the "SKIP" key on the operation panel is valid, i.e. the skip character in the block is invalid;

N△△△△△: block number. A block can have or not it; number △△△△△ following N is 5-digit positive integer 00001～99999, and the system alarms when the input number is decimal.

Countless words: one block can input countless words, and one block can have one or more words or have no words.

，： “EOB” is a end character when one block is completed, “;” is displayed in LCD, there must be have one end character for one block;

2. Format requirements

（1）In one block, there can be no blank space between block number and word, and can be countless blank space(the total characters of one block is within 255);

（2）In one block, there can be not or be countless space between skip character and block number or words;

（3）In one block, there can be not or be countless space between end character of block and its front word or blocks;

Each block can be up to 255 characters, including skip character, block number, command, space, end character of block “;”;

（4）The system automatically ignores the content with small bracket “（”,“）”.

Explanations of program annotation:

Note: The annotation of program home as the total annotation of a program is displayed in the program catalog window, the created program automatically creates the small brackets “（”、“）”, if they are deleted, the system has no them and they can be replaced by “;”.

3．Parameters related block number:

（1）whether the system automatically creates block number or not:

User can set whether the system automatically creates block number or not in editing program by setting Bit 5(SEQ) of NO.0000;

（2）Use can set the interval value in automatically creating block number by setting NO.3216.

Note: Sprit(/) explanations:

1. When the sprit (/) is used to skip character, it is generally placed the beginning of block, otherwise , and the messages from the sprit to EOB code are ignored. For example: U10.G00/04; when the skip function is started, the system executes U10. G00;(G00 U10.), when it stops, the system executes U10. G0004;(G04 U10.);

2. For cycle command buffer, when a block reads from memory to buffer memory, whether the skip function is valid or not has been executed. After a block reads into buffer memory, i.e. the system changes skip switch state, but does not influence the block which has read into the buffer memory;

3. Sprit (/) (closed in bracket[]) and sprit(/) right to value statement “=” in <Expression> are taken as division operation character instead of skip character.

1.5.3 Word

1. Format: address + number. There must not be space between address and number.

Presently, the system permissively input addresses: G, M, S, T, F, X, Y, Z, U, V, W, P, Q, I, J, K, R, L, A, B, C, H , N, O， and will add other;

Command number range following address is referred to the following table.

Table 1-5-1 word table

Address	Function	mm input	inch input	Related G codes
O	Program name	0～9999	0～9999	
N	Line label	1～99999	1～99999	
G	Preparatory function	See G code	See G code	

M	Miscellaneous function	0～9999	0～9999	
S	Spindle speed	（G96） 0～20000 m/min	（G96） 0～2000 feet/min	
		（G97） 0～20000 r/min	（G97） 0～20000 r/min	
T	Tool offset	0000～9999	0000～9999	
F	Feedrate per minute	（ISB system） 1 ～60000 mm/min	（ISB system） 0.01～2400 inch/min	G98
		（ISC system） 1 ～24000 mm/min	（ISC system） 0.01～960 inch/min	
	Feedrate per rev	（ISB system） 0.01～500mm/r	（ISB system） 0.01～9.99inch/r	G99
		（ISC system） 0.01～500mm/r	（ISC system） 0.01～9.99 inch/r	
	Pitch	0.01～500 mm	0.01～9.99inch	Relative commands for thread machining
X	X absolute coordinate value((linear axis), delay time (*1)	（ISB system） -99999.999～99999.999mm	（ISB system） -9999.9999～9999.9999inch	Relative command of axis, G04
		（ISC system） -9999.9999～9999.9999mm	-999.99999～999.99999inch	
Y	Y absolute coordinate value(linear axis) (*1)	（ISB system） -99999.999～99999.999 mm	（ISB system） -9999.9999～9999.9999 inch	Relative command of axis
		（ISC system） -9999.9999～9999.9999 mm	（ISC system） -999.99999～999.99999 inch	
Z	Z absolute coordinate value (linear axis) (*1)	（ISB system） -99999.999～99999.999 mm	（ISB system） -9999.9999～9999.9999 inch	Relative command of axis
		（ISC system） -9999.9999～9999.9999 mm	（ISC system） -999.99999～999.99999 inch	
A	A absolute coordinate value(linear axis) (*1)	（ISB system） -99999.999～99999.999 mm	（ISB system） -9999.9999～9999.9999 inch	Relative command of axis
		（ISC system） -9999.9999～9999.9999 mm	（ISC system） -999.99999～999.99999 inch	
B	B absolute coordinate value(linear axis) (*1)	（ISB system） -99999.999～99999.999 mm	（ISB system） -9999.9999～9999.9999 inch	Relative command of axis
		（ISC system） -9999.9999～9999.9999 mm	（ISC system） -999.99999～999.99999 inch	
C	C absolute coordinate value (rotary axis) (*1)	（ISB system） -99999.999～99999.999 deg	（ISB system） -99999.999～99999.999 deg	Relative command of axis
		（ISC system） -9999.9999～9999.9999 deg	（ISC system） -9999.9999～9999.9999 deg	
U	X relative coordinate value, finishing allowance in G71, G72, G73, X tool retraction distance and specified delay time(*1) in G73, (*1)	（ISB system） -99999.999～99999.999 mm	（ISB system） -9999.9999～9999.9999 inch	Relative command of axis,G71,G72,G73,G04
		（ISC system） -9999.9999～9999.9999 mm	（ISC system） -999.99999～999.99999 inch	

	Cut depth in G71(modify parameter manual) (*2)	（ISB system） 0.001~99999.999 mm	（ISB system） 0.0001~9999.9999 inch	G71
		（ISC system） 0.0001~9999.9999 mm	（ISC system） 0.00001~999.99999 inch	
V	Y relative coordinate value(linear axis) (*1)	（ISB system） -99999.999～99999.999 mm	（ISB system） -9999.9999～9999.9999 inch	Relative command of axis
		（ISC system） -9999.9999～9999.9999 mm	（ISC system） -999.99999～999.99999 inch	
W	Z relative coordinate value, Z finishing allowance in G71, G72, G73, Z tool retraction distance (*1) in G73 (*1)	（ISB system） -99999.999～99999.999 mm	（ISB system） -9999.9999～9999.9999 inch	Relative command of axis, G71, G72, G73,
		（ISC system） -9999.9999～9999.9999 mm	（ISC system） -999.99999～999.99999 inch	
	Cut depth (*2) in G72 (*2)	（ISB system） 0.001~99999.999 mm	（ISB system） 0.0001~9999.9999 inch	G72
		（ISC system） 0.0001~9999.9999 mm	0.00001~999.99999 inch	
R	Arc radius (*1)	（ISB system） -99999.999～99999.999 mm	（ISB system） -9999.9999～9999.9999 inch	G02,G03
		（ISC system） -9999.9999～9999.9999 mm	（ISC system） -999.99999～999.99999 inch	
	Taper and thread taper (*1) in G90, G92, G94, G76 (*1)	（ISB system） -99999.999～99999.999 mm	（ISB system） -9999.9999～9999.9999 inch	G90,G92,G94,G76
		（ISC system） -9999.9999～9999.9999 mm	（ISC system） -999.99999～999.99999 inch	
	Tool retraction（*2） in G71,G72 （*2）	（ISB system） 0～99999.999 mm	（ISB system） 0～9999.9999 inch	G71,G72
		（ISC system） 0～9999.9999 mm	（ISC system） 0～999.99999 inch	
	Roughing times in G73	1～999（times）	1～999（times）	G73
	Thread increment in variable pitch cutting	0.01～500.000 mm -0.01～-500.000 mm	0.01～9.99inch -0.01～-9.99inch	G34
	Tool retract movement after cutting in G74, G75 and tool retraction after cutting to end point （*2）	（ISB system） 0～99999.999 mm	（ISB system） 0～9999.9999 inch	G74,G75
		（ISC system） 0～9999.9999 mm	（ISC system） 0～999.99999 inch	
	Finishing amount （*2） in G76	（ISB system） 0.001～99999.999 mm	（ISB system） 0.0001～9999.9999 inch	G76
		（ISC system） 0.0001～9999.9999 mm	（ISC system） 0.00001～999.99999 inch	
P	Dwell time	0~99999999ms	0~99999999 ms	G04
	G30 returning to No.n reference position	2,3,4	2,3,4	G30（default to 2）
	Commands for macro program number, subprogram and subprogram call times	1～9999	1～9999	G65,G66,M98 （ default times is 1）

	Line number assignment in G70, G71, G72,G73	0～99999	0～99999	G70,G71,G72,G73
	X cycle movement（*3） in G74, G75	0 ～ 99999999 × least command unit	0～99999999×least command unit	G74,G75
	Thread cutting parameter in G76	Including 3 parameters: Thread finishing times：1～99 Thread run-out length：00～99（*0.1 pitch） Angle between two teeth：0°~99°	Including 3 parameters: Thread finishing times：1～99 Thread run-out length：00～99（*0.1 pitch） Angle between two teeth：0°~99°	G76
	Thread tooth height （*3） in G76	1 ～ 99999999 × least command unit	1～99999999×least command unit	G76
Q	Line number assignment in G70, G71, G72, G73	0～99999	0～99999	G70,G71,G72,G73
	Tool infeed amount(*3) in Z brokenly infeed in G74,G75	0 ～ 99999999 × least command unit	0～99999999×least command unit	G74,G75
	Min. cutting amount (*3) in G76 thread roughing	0 ～ 99999999 × least command unit	0～99999999×least command unit	G76
	1st thread cutting depth （*3） in G76 thread roughing	1 ～ 99999999 × least command unit	1～99999999×least command unit	G76
	Initial angle （*3）of 1st circle in thread cutting （*3）	0 ～ 99999999 × least command unit （default to 0）	0～99999999×least command unit （default to 0）	G32,G34,G92
L	Macro program call times assignment	1～9999（default to 1）	1～9999（default to 1）	G65,G66
	Head quality of multi-thread	1～99（default to 1）	1～99（default to 1）	G92
I	Relative starting point of arc center is in X vector (*1)	（ISB system） -99999.999～99999.999mm	（ISB system） -9999.9999～9999.9999 inch	G02,G03
		（ISC system） -9999.9999～9999.9999 mm	（ISC system） -999.99999～999.99999 inch	
J	Relative starting point of arc center is in Y vector (*1)	（ISB system） -99999.999～99999.999 mm	（ISB system） -9999.9999～9999.9999 inch	G02,G03
		（ISC system） -9999.9999～9999.9999 mm	（ISC system） -999.99999～999.99999 inch	
	Movement in short axis when thread run-out is executed (*1)	（ISB system） -99999.999～99999.999 mm	（ISB system） -9999.9999～9999.9999 inch	G32,G34,G92
		（ISC system） -9999.9999～9999.9999 mm	（ISC system） -999.99999～999.99999 inch	
K	Relative starting point of arc center is in Z vector (*1)	（ISB system） -99999.999～99999.999 mm	（ISB system） -9999.9999～9999.9999 inch	G02,G03
		（ISC system） -9999.9999～9999.9999 mm	（ISC system） -999.99999～999.99999 inch	

	Length in long axis when thread run-out is executed （*2）		（ISB system） 0～99999.999 mm （ISC system） 0～9999.9999 mm	（ISB system） 0～9999.9999 inch （ISC system） 0～999.99999 inch	G32,G34,G92
IF		Conditional judgement			
THEN	TH	Execution after IF conditional judgment is completed successfully			
GOTO	GO	Non-conditional skip			
WHILE	WH	Cycle judgment			
DO		Start to execute cycle			
END	EN	Return to WHILE			
EQ	==	Equal to	Judgement logic is used to brackets following IF, WHILE		
NE	<>	Not equal to			
GT	>	Greater than			
GE	>=	Greater than or equal to			
LT	<	Less than			
LE	<=	Less than or equal to			
SIN	SI	Sine	Functional function is used to count expression value		
ASIN	AS	Anti-sine			
COS	CO	Cosine			
ACOS	AC	Anti-cosine			
TAN	TA	Tangent			
ATAN	AT	Anti-tangent			
SQRT	SQ	Square root			
ABS	AB	Absolute value			
ROUN	RO	Rounding-off			
FIX	FI	Down integer			
FUP	FU	Up integer			
LN		Nature logarithm			
EXP	EX	Exponential function			
OR		OR			
XOR	XO	OR AND			
AND	AN	AND			
BIN	BI	Converse from BCD to BIN			
BCD	BC	Converse from BIN to BCD			
123456789	With to compose the value of word, the leading 0 can be omitted				
0	Word is 0 and is different with Null value				
+	Number count and number expression				

-		
*		
/	Skip command, selectively skip the commands following the character	
.	Floating point number with number	
=	Variable assignment	
[	Prior operation of expression and conditional judgement prompt	
]		
#	Variable	
;	End of program in the block, following annotation	
(	Annotation start in the block. Example: (X20.)W-10.; not execute X20.	
)	Annotation end in the block	
%	End of program	

Note 1: The 2-digit following the decimal point of F value is value, and the more following the two-digit is ignored.

Note 2: The expression can follow the word, the value counted by the expression is taken as the value of the word, and the expression should have[] , and there must not be the space between the word and the expression. For example X[#1-#110] Z[#1+SIN[#120]].

*1）: When the address values in the above table, X, Y, Z, C, A, B, C, U, V, W, H, I, J, K, R are taken as word address, their value ranges are controlled by the following parameters:

（1）No.0000#2 INI

INI input unit

0：metric

1：inch

（2）No.1006#0 ROTx

ROTx set linear axis or rotary axis

0：linear axis

1：rotary axis

（3）No.0004#1 ISC

ISC set least input unit and least command increment

Table 1-5-2 set least input unit and least command increment

ISC	Least setting unit	For short
0	0.001mm, 0.001deg or 0.0001inch	IS-B
1	0.0001mm, 0.0001deg or 0.00001inch	IS-C

Table 1-5-3 least command unit and value range

Address	Parameter setting			Least command unit	Range
X,Y,Z,C,A,B,C,U,V,W,H	ROTx=0 Rotary axis	Rotary axis is not	ISC=0 ISB	0.001deg	-99999.999～ 99999.999 deg

		related to INI	ISC=1 ISC	0.0001deg	-9999.9999～ 9999.9999 deg
X,Y,Z,C,A,B,C,U,V, W,H,I,J,K,R	ROTx=1 Linear axis	INI=0 Metric	ISC=0 ISB	0.001mm	-99999.999～ 99999.999 mm
			ISC=1 ISC	0.0001mm	-9999.9999～ 9999.9999 mm
		INI=1 Inch	ISC=0 ISB	0.0001inch	-9999.9999～ 9999.9999 inch
			ISC=1 ISC	0.00001inch	-999.99999～ 999.99999 inch

When these word addresses follow data, data precision is least command unit, and excessive data is ignored. When a word address follows variable number or has [] expression, the word value has decimal data, and its precision is the least command unit, but its excessive data rounds.

（4）No.3401#0 DPI

DPI can use decimal address. When the decimal is omitted, its setting is as follows:

0：least setting unit

1：unit: mm，inch，sec

When parameter DPI is set to 1, word range is referred to Table 1-5-3;

When DPI is set to 0, and word omits its decimal, its value range is -99999999～99999999, data unit is the least command unit in Table 1-5-3.

*2）: Command value calculation method specified by U, W, R, K is the same that of *1）, they meet the value range described in *1） and limit value range according to preparatory function.

*3）: Position specified value commanded by P, Q is 0～99999999, data unit is the least command unit in Table 1-5-3. value range is limit by specific preparatory function.

2. Word value and state will change when the system runs, the following table separately explains each word omit and state when the system is ON, resets.

Table 1-5-4 word state

Character	Function	Initial value when power-on	Default value	Keep in the next block?	Value after pressing reset key	Related explanation
O	Program name	Value reserved by last power-on	Current value	Yes	Yes	None
G	Preparatory function	Initial mode in each group	Modal value	No	parameter （ CLR ） NO.3402#6	None
M	Miscellaneous function M00, M01, M02, M30, M98, M99		Current value	No（function reserved）	Current value	Specified by PLC, set by parameter
S	Analog spindle speed	0	Current value	Yes	Current value，output is invalid	
T	Tool offset	Value reserved by last power-on	Current value	Yes	Current value	

F	Feedrate per minute	Parameter value	Current value	Yes	parameter（CLR）NO.3402#6	
	Feedrate per rev	Null	Current value	Yes	Current value	
	Pitch	Null	Current value	Yes	Current value	
X	Delay time	Null	0	No	0	
	X absolute coordinate value	0	Current value	Yes	Current value	
Y	Y absolute coordinate value	0	Current value	Yes	Current value	
Z	Z absolute coordinate value	0	Current value	Yes	Current value	
C	C absolute coordinate value	0	Current value	Yes	Current value	
U	Delay time	Null	0	No	Null	
	X relative coordinate value	0	0	No	Current value	
	X allowance in finishing	Null	0	No	Null	
	Cutting depth in G71	Parameter value	Parameter value	Yes	Parameter value	
V	Y relative coordinate value	0	0	No	Current value	
W	Z relative coordinate value	0	0	No	Current value	
	Z allowance in finishing	空	0	No	Null	
	Cutting depth in G72	Parameter value	Parameter value	Yes	Parameter value	
H	C increment value	0	0	No	Current value	G00
		0	0	No	Current value	Polar coordinate interpolation
R	Arc radius	0	0	No	Current value	
	Taper G90, G92, G94 and thread taper	0	0	Yes	Current value	
	Tool retraction in G71, G72	Parameter value	Parameter value	Yes	Parameter value	
	Roughing times in G73	Parameter value	Parameter value	Yes	Parameter value	
	Clearance in G74,G75	Parameter value	Parameter value	Yes	Parameter value	
	Clearance to end point in G74,G75	0	0	No	Null	

	Finishing cutting amount in G76	Parameter value	Parameter value	Yes	Parameter value	
P	Dwell time	Null	0	No	Null	
	G30 returning to No. n reference position	Null	2	No	Null	
	Macro program number, subprogram, subprogram call times	Null	Alarm	No	Null	
	Line assignment in G70, G71, G72, G73	Null	Alarm	No	Null	
	X cycle movement in G74,G75	Null	0	No	Null	
	Thread cutting in G76	Parameter value	Parameter value	Yes	Parameter value	
	Thread tooth height in G76	0	Alarm	No	Null	
Q	Line assignment in G70, G71, G72, G73	Null	Alarm	No	Null	
	Z broken tool infeed amount in G74, G75	Null	0	No	Null	
	Least cutting amount in G76 roughing	Parameter value	Parameter value	Yes	Parameter value	
	1st thread cutting depth in G76 thread roughing	Null	Alarm	No	Null	
	1st circle start angle in thread cutting	Null	0	No	0	
	Check offset in spindle fluctuation check	Null	0	No（the parameter cannot be modified	0	
L	Macro program call times assignment	1	1	No	Null	
I	X vector of circle center corresponding to starting point	0	0	No	Current value	
	X calculation direction in cancelling radius compensation	Null	Null	No	Null	
J	Y vector of circle center corresponding to starting point	0	0	No	Current value	
	Y calculation direction in cancelling radius compensation	Null	Null	No	Null	
K	Z vector of circle center corresponding	0	0	No	Current value	

	to starting point					
	Pitch increment in variable pitch thread cutting	Null	0	Yes	Current value	
	X travel lower limit value	Null	Alarm	No	Current value	
	Z calculation direction in cancelling radius compensation	Null	Null	No	Null	

1.5.4 Block number

Format: N △△△△△

△△△△△ is 5-digit integer 00001～99999, and its leading zero can be omitted.

（1）Can or not input a block number in one block(must input block number in target block in which program skips), when many block number are input in one block, only the last block number is valid;

（2）Block number can be placed any position of block but it is suggested that it should be placed at the initial position in order to search and read；

（3）There can be many same block number in one program, but the block number of target block of program skip has only one; otherwise, the program skips to the nearest block to the block;

（4）block number can be placed at will.(it is suggested that it should be placed by the rising or falling monotonously;

Note: When the block number exceeds the range, and the program runs or the grammatical check is done, the relevant alarm occurs.

1.5.5 Main program and subprogram

To simply the programming, when the same or similar machining path and control procedure is used many times, its program commands are edited to a sole program to call. The main program is defined to call others and the subprogram is to be called. They both take up the program capacity and storage space of system. The subprogram has own name, and can be called at will by the main program and also can run separately. The system returns to the main program to continue when the subprogram ends as follows:

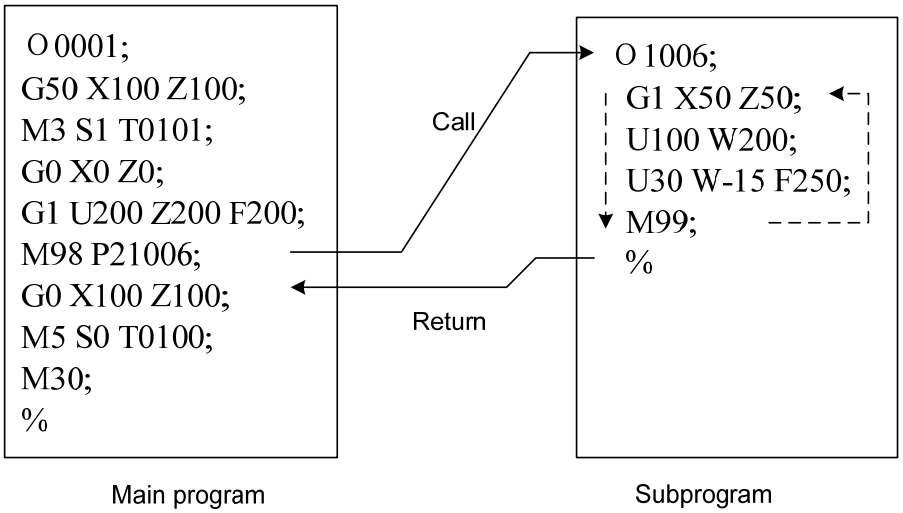


Fig.1-11

1.6 Program Run

1.6.1 Sequence of program run

Running the current open program must be in Auto mode. GSK988T cannot open two or more programs at the same, and runs only program any time. When the first block is open, the cursor is located in the heading of the first block and can be moved in Edit mode. In the run stop state in Auto mode, the program starts to run by the cycle start signal (CYCLE START key is pressed or external cycle start signal)from a block pointed by current cursor, usually blocks are executed one by one according to their programming sequence, the program stops running till executing M02 or M30. The cursor moves along with program running and is located at the heading of the current block. Sequence and state of program running are changed in the followings:

- The program stops run after pressing RESET or EMERGENCY STOP button;
- The program stops running when the system or PLC alarms;
- The program runs and single block stops (the program run stops after the current block runs completely) in Edit, MDI mode, and then a block pointed by the current cursor starts running after the system switches into Auto mode, the CYCLE START key is pressed or external cycle start signal is switched on;
- The program stops run in Manual(Jog), Handwheel (MPG), Single Block, Program Reference position Return, Machine Reference position Return mode and it continuously runs from current position after the system is switched into Auto mode and the CYCLE START key is pressed or the external cycle start signal is switched on;
- The program pauses after pressing the FEED HOLD key or the external cycle start signal is switched off, and it continuously runs from current position after pressing the CYCLE START key or the external cycle start signal is switched on;
- When Single Block is ON, the program pauses after every block is executed completely, and then it continuously runs from the next block after the CYCLE START key is pressed or the external cycle start signal is switched on;
- Block with “/” in the front of it is not executed when the block skipping switch is ON;
- The system skips to the target block to run after executing G65;
- Please see Section Three G Commands about execution sequence of G70~73;

- Call corresponding subprograms or macro program to run when executing M98 or M9000~M9999; the system returns to main program to call the next block when executing M99(if M99 specifies a target block number, the system returns to it to run) after the subprograms or macro programs run completely;
- The system returns to the first block to run and the current program is executed repetitively when M99 is executed in a main program.

1.6.2 Execution sequence of word

There are many words (G, X, Z, F, R, M, S, T and so on) and most of M, S, T is transmitted to PLC by NC explaining and others are directly executed by NC. M98, M99, M9000～M9999, S word for specifying spindle speed (r/min, m/min) is directly executed by NC.

NC firstly executes G and then M commands when G codes and M00, M01, M02 and M30 are in the same block.

NC firstly executes G and then M commands(without transmitting M signal to PLC) when G codes and M98, M99, M9000～M9999 are in the same block.

When G codes and M, S, T executed by PLC are in the same block, PLC defines M, S, T and G to be executed simultaneously, or execute M, S ,T after G codes. Please see User Manual of machine manufacturer for execution sequence of commands.

Execution sequence of G, M (except for the above M codes), S, T defined by GSK988T PLC in the same block is determined by PLC, which is divided into two methods:

a) Movement codes and M miscellaneous code are executed simultaneously.

b) Execute miscellaneous codes after executing movement codes.

Refer to the machine manufacture's user manual for the concrete execution method.

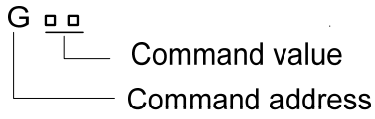
The second method is executed when there is M9, M99, M13, M33 or M5 for our GSK ladder.

Note: When G28 or G30 and M01 are in the same block, the pause after zero return is done. When there is a single block stop command without M01,the single block stop is executed at the middle point and zero return completion position. When G28 or G30 and M01 are in the same block and the single block stop is valid, the pause is executed after zero return.

Chapter II G Commands

2.1 Summary

G command consists of command address G and its following command value, used for defining the motion mode of tool relative to the workpiece, defining the coordinates and so on. Refer to G commands as Fig. 2-1.



Note 1: The leading zero of the command value can be omitted. Example: G02 is equivalent to G2, G01 to G1.

Note 2: The leading zero following the command value cannot be omitted. Example: G20 and G2 are different G commands in the different group; G12.1 is legal but G12.10 is illegal.

Note 3: The command value can be up to 8-bit digit. Example: G00000002 is correct and valid, equivalent to G02.

Note 4: Except for G12.1 and G13.1, other G command cannot be with the decimal point, otherwise, the alarm occurs. For example: G20.0, G00.0, G18.are illegal.

2.1.1 G command classification

G commands are divided into: modal G command and non-modal G command.

After a G command is executed, its defined function or state remains valid till other G command is specified in the same group, this G command is called the modal. After the modal G command is executed, before its defined function or state is changed, the G command cannot be input again when the following block executes the G word.

After a G command is executed, its defined function or state is valid once, its word must be input again when it is executed, and so the G command is called the non-modal.

Example 1: G01 and G00 are modal.

```
G01 X_;
    Z_;          } G01 is valid in the range
    X_;
G00 Z_;
    X_;          } G00 is valid in the range
G01 X_;
```

Example 2: G04 is non-modal.

```
O0002;
    G0 X50 Z5;          (Rapid traverse to X50 Z5)
    G04 X4;             (Delay 4s)
    G04 X5;             (Delay 5s again，G04 is non-modal and must be input again)
    M30;
```

Table 2-1 G command list

G command	Group	Function	Classification
*G00	01	Positioning(rapid traverse)	Modal

G01		Linear interpolation	
G02		Circular interpolation(CW)	
G03		Circular interpolation(CCW)	
G04	00	dwell	Non-modal
G7.1 (G107)		Cylindrical interpolation	
G10		Programmable data input	
G11		Programmable data input cancel	
G12.1 （G112）	21	Polar coordinate interpolation mode	Modal
***G13.1** （G113）		Polar coordinate interpolation mode cancel	
G17	16	XpYp level selection	Modal
***G18**		ZpXp level selection	
G19		YpZp level selection	
G20	06	Inch input	Modal
***G21**		mm input	
***G22**	09	Stored travel check ON	Modal
G23		Stored travel check OFF	
G28	00	Return to reference position	Non-modal
G30		Return to 2nd, 3rd, 4th reference position	
G32	01	Constant pitch thread cutting	Modal
G34		Variable pitch thread cutting	
***G40**	07	Tool radius compensation cancel	Modal
G41		Cutter compensation left	
G42		Cutter compensation right	
G50	00	Workpiece setting or max. spindle speed setting	Non-modal
G52		Local coordinate system setting	
G53		Machine coordinate system setting	
***G54**	14	Select workpiece coordinate system 1	Modal
G55		Select workpiece coordinate system 2	
G56		Select workpiece coordinate system 3	
G57		Select workpiece coordinate system 4	
G58		Select workpiece coordinate system 5	
G59		Select workpiece coordinate system 6	
G61	15	Exact stop mode	Modal
***G64**		Cutting mode	
G65	00	Non-modal macro program call	Non-modal
G66	12	Macro program mode call	Modal
***G67**		Cancel macro program mode call	
G70	00	Finishing cycle	Non-modal
G71		Axial roughing cycle	
G72		Radial roughing cycle	
G73		Closed cutting cycle	
G74		Axial grooving cycle	

G75		Radial cutting multi-cycle	
G76		Multi thread cutting cycle	
*G80	10	Cancel drilling fixed cycle	Modal
G83		End drilling cycle	
G84		End rigid/common tapping cycle	
G85		End boring cycle	
G87		Side drilling cycle	
G88		Side rigid/common tapping cycle	
G89		Side boring cycle	
G90	01	Axial cutting cycle	Modal
G92		Thread cutting cycle	
G94		Radial cutting cycle	
G96	02	Constant surface speed control	Modal
***G97**		Constant speed control	
***G98**	05	Feed per minute	Modal
G99		Feed per revolution	

Note 1: G commands in Group 01, 05, 09 separately set their state in No.3402 Bit0(G01), Bit4 (FPM), Bit7 (G23) when the system is power-on, the G commands in Group 06 in No.0000 Bit2(INI); when the system is turned on, the modal G command in other groups are at the state designated by *.

Note 2: When the system resets, No.3402 Bit6 (CLR) is set to 0, the modal of the G command remains unchanged; when it is set to 1, the modal is changed to the one which is at the power-on, but G22 and G23 in Group 09 and G20 and G21 in Group 06 remain unchanged.

Note 3: G commands in Group 00 are non-modal.

Note 4: G commands in Group 00 and ones in Group 01 are specified in the same block, G commands in Group 00 are valid, G commands in Group 01 only change their modal.

Note 5: Commands in Group 06, 09, 21 and ones in other groups cannot be in the same block, commands in Group 12 and G65 are specified only in a separate block.

Note 6: When No.3403 Bit6(AD2) is set to 0, many G commands in the different groups can be specified in the same block, and the G command specified at last is valid; when it is set to 11, the alarm occurs.

Note 7: When compiling a G command in one block needs a word, and the compiled cannot use the word, the word is ignored(for example: G00 X_ Z_ R_ ， R_ is ignored); when the ignored word format is not correct, the alarm occurs (For example: G00 X_ Z_ R2.3.1）.

Note 8: When compiling No.1020 does not have the axis word including the absolute address or incremental address, the alarm occurs.

2.1.2 Omitting word input

To simplify the programming, their command values are reserved after executing words in Table 2-2. If the words are contained in the previous blocks, they cannot be input when the words are used with the same values and definitions in the following blocks.

Table 2-2

Command address	Function	Initial value when power-on
U	Cutting depth in G71	№51 parameter value
U	Move distance of X tool retraction in G73	№53 parameter value
W	Cutting depth in G72	№51 parameter value
W	Move distance of X tool retraction in G73	№54 parameter value
R	Move distance of tool retraction in G71, G72 cycle	№52 parameter value
R	Cycle times of stock removal in turning in G73	№55 parameter value
R	Move distance of tool retraction after cutting in G74, G75	№56 parameter value
R	Allowance of finishing in G76	№60 parameter value
R	Taper in G90, G92, G94, G96	0
(G98) F	Feedrate per minute(G98)	№30 parameter value
(G99) F	Feedrate per rev (G99)	0
F	Metric pitch(G32, G92, G76)	0
I	Inch pitch(G32, G92)	0
S	Spindle speed specified(G97)	0
S	Spindle surface speed specified(G96)	0
S	Spindle speed switching value output	0
P	Finishing times of thread cutting in G76; Tool retraction width of thread cutting in G76 Angle of tool nose of thread cutting in G76;	№57 parameter value №19 parameter value №58 parameter value
Q	Min. cutting value in G76	№59 parameter value

Note 1: For the command addresses with functions (such as F, used for feedrate per minute, feedrate per rev and metric pitch and so on), they can be omitted not to input when executing the same function to definite words after the words are executed. For example, after executing G98 F_ without executing the thread command, the pitch must be input with F word when machining metric thread.

Note 2: When the words in the aboved table (except for F, S) are not omitted, the input new command value is written to the corresponding parameter.

Note 3: When X（U）, Y（V）, Z（W）, A, B or C（H）are used to the end point coordinates of the specified block and their words in the block are not input, the system takes the absolute coordinates of the current X, Y, Z, A, B or C as the coordinates of the end point.

Example 1:（run after the first power-on）:

```
O0003;
    G98 F500 G01 X100 Z100;          （G98: feed/minute，500mm/min）
    G92 X50 W-20 F2 ;                （thread cutting, F must be input when it is the pitch）
    G99 G01 U10 F0.01 ;              （G99: feed/minute, F is input again）
    G00 X80 Z50 ;
    M30;
```

Example 2:

```
O0001;
    G0 X100 Z100;        （rapidly traverse to X100 Z100; the modal G0 is valid）
```

X20 Z30; （rapidly traverse to X20 Z30; the modal G0 can be omitted）

G1 X50 Z50 F300; （linear interpolation to X50 Z50， 300mm/min; the modal G1 is valid）

X100; （linear interpolation to X100 Z50，300mm/min; When Z coordinate is not input, the current coordinate value Z50 is used; F300 is kept, G01 can be omitted when it is modal.）

G0 X0 Z0; （rapidly traverse to X0 Z0，the modal G0 is valid）

M30;

2.1.3 Related definitions

Definitions of word are as follows except for the especial explanations:

Starting point: position before the current block runs;

End point: position after the current block ends;

X: X absolute coordinates of end point;

Xp：absolute coordinate of X end point or one which is parallel to X;

U: different value of X absolute coordinate between starting point and end point;

Y：Y absolute coordinate of end point;

Yp：absolute coordinate of Y end point or one which is parallel to Y;

V：different value of Y absolute coordinate;

Z: Z absolute coordinates of end point;

Zp：absolute coordinate of Z end point or one which is parallel to Z;

W: different value of absolute coordinates between starting point and end point;

C：C absolute coordinate of end point;

H：different value of C absolute coordinate between end point and starting point;

A：A absolute coordinate of end point;

B：B absolute coordinate of end point;

F：cutting feedrate.

IP：it is the combination of axes to execute the data provided by G command, the later specified address is valid when the absolute address and relative address of one axis are defined and are in the same block to be edit. The range for each axis in corresponding parameter is as follows:

2.2 Rapid Traverse (Positioning) G00

Command function: In the absolute command, the tool rapidly traverses to the position specified by the workpiece coordinate system; in the incremental command, the tool rapidly traverses to the position which offsets the specified value of the current position.

Command format：G00 IP__；

Command explanation: IP: it is the end point coordinate value of the tool traversing for the absolute command; it is the tool traversing distance for the incremental command.

Command path:

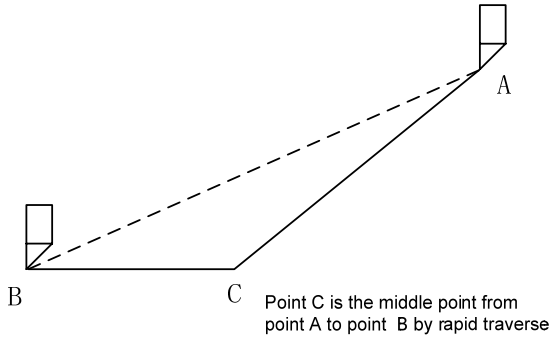
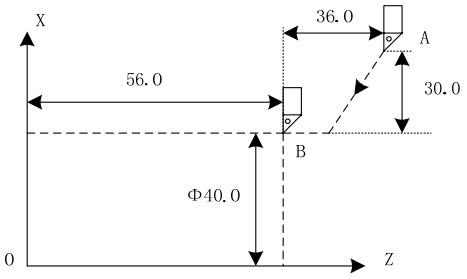


Fig. 2-1 rapid traverse(positioning)

Execution process:



Program：（Diameter programming）
G00 X40.0 Z56.0；（Absolute programming）or
G00 U60.0 W-36.0；（Incremental programming）or
G00 X40.0 W-36.0；（Compound programming）or
G00 U60.0 Z56.0；（Compound programming）

Fig. 2-2 positioning example

Note 1: The rapid traverse speed(G00) is set in No.1420 and is not related to the commanded feedrate F value in the block.

Note 2: Whether the initial mode of Group 01 when power-on is G00 or G01 is determined by No.3402 Bit0(G01).

2.3 Linear Interpolation G01

Command function: the tool executes the linear traverse.

Command format: G01 IP__ F__; it can be omitted to G1

Command explanation： IP_: it is the end point coordinate value of tool traversing for the absolute command; it is the tool traversing distance for the incremental command.

F_: it is the feedrate of the tool and its ranges is shown below.

Feed mode		Metric（mm）input	Inch (inch) input
G98	ISB system	1～60000 mm/min	0.01～2400 inch/min
	ISCsystem	1～24000 mm/min	0.01～960 inch/min
G99	ISB system	0.01～500mm/r	0.01～9.99inch/r
	ISC system	0.01～500mm/r	0.01～9.99 inch/r

Command path:

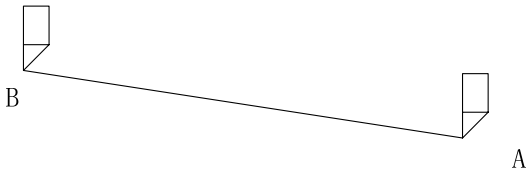
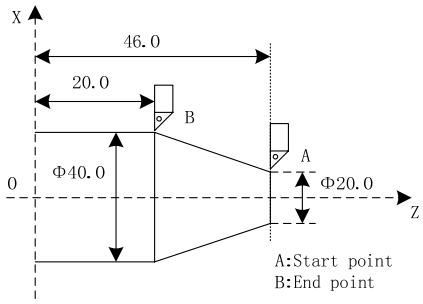


Fig. 2-3 linear interpolation

Execution process：



Programming：（Diameter programming）
G01 X40.0 Z20.0 F500;（Absolute programming） or
G01 U20.0 W-26.0；（Incremental programming）or
G01 X40.0 W-26.0；（Compound programming）or
G01 U20.0 Z20.0；（Compound programming）

Fig. 2-4 Linear interpolation example

Note 1: The tool traverses to the specified position along the linear at the speed specified by F. Before the new value is specified, each program is not needed to specify.

Note 2: The actual cutting feedrate is the product between the feedrate override and F command value.

Note 3: The actual cutting feedrate is limited by max. cutting feedrate MFR of No. 1422.

Note 4: G04 supports the synchronous interpolation of linear axis and rotary axis. The command speed includes the speed of rotary axis. When there is only the combination speed of linear axis, the display value of actual speed does not include the actual speed of rotary axis.

2.4 Arc Interpolation G02, G03

Command function: The tool traverses along an arc on the specified level.

Command format：

$$G17\left\{\begin{matrix}G02\\G03\end{matrix}\right\}Xp\_Yp\_\left\{\begin{matrix}R\_\\I\_J\_\end{matrix}\right\}F\_$$

$$G18\left\{\begin{matrix}G02\\G03\end{matrix}\right\}Xp\_Zp\_\left\{\begin{matrix}R\_\\I\_K\_\end{matrix}\right\}F\_$$

$$G19\left\{\begin{matrix}G02\\G03\end{matrix}\right\}Yp\_Zp\_\left\{\begin{matrix}R\_\\J\_K\_\end{matrix}\right\}F\_$$

Command explanations：

Command	Description
G17	XpYp level selection
G18	ZpXp level selection
G19	YpZp level selection
G02	Arc interpolation (CW)
G03	Arc interpolation (CCW)
Xp_	Movement of X or an axis parallel to it (set by No.1022)
Yp_	Movement of Y or an axis parallel to it (set by No.1022)
Zp_	Movement of Z or an axis parallel to it (set by No.1022)

I_	Distance between starting point of Xp axis to center of arc (with sign, its range referred to the following table)
J_	Distance between starting point of Yp axis to center of arc (with sign, its range referred to the following table)
K_	Distance between starting point of Zp axis to center of arc (with sign, its range referred to the following table)
R_	Arc radius (with sign, it is the radius value when machining, range referred to the following table)
F_	Feedrate along arc（its range is the same that of G01）

Address	Incremental system	Metric input（mm）	Inch input (inch)
I, J, K, R	ISB system	-99999.999～99999.999	-9999.9999～9999.9999
	ISC system	-9999.9999～9999.9999	-999.99999～999.99999

I, J, K have sign symbols according their directions, they are positive when their directions are the same those of Xp, Yp, Zp, otherwise, they are negative.

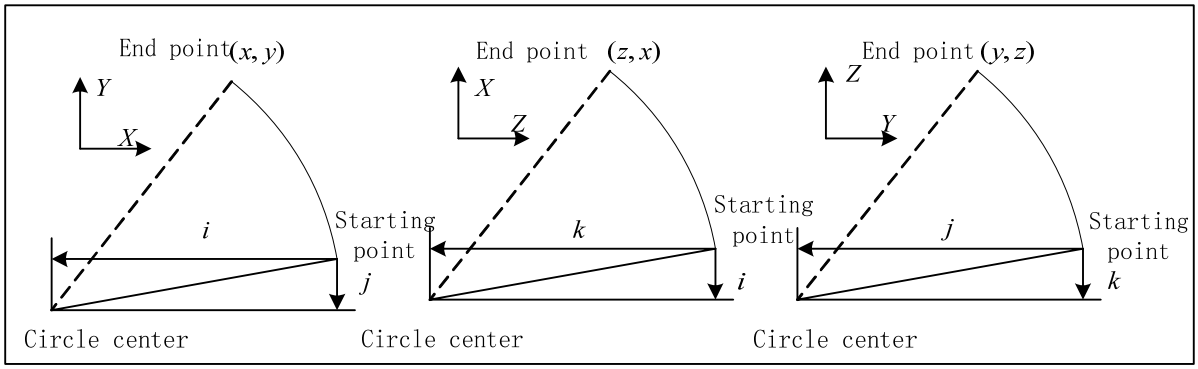


Fig. 2-5

Command path（arc direction）:

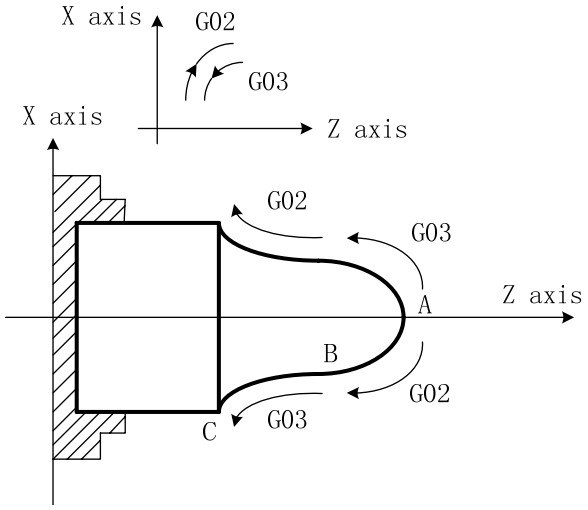
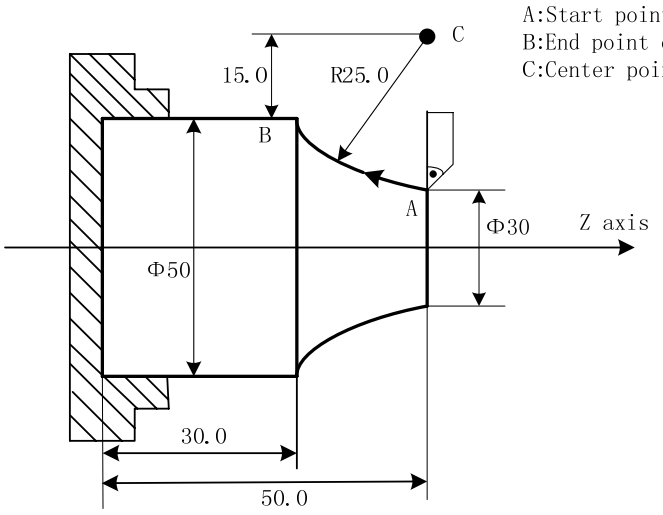


Fig. 2-6 Arc interpolation

Execution process：（taking G02 as an example）



Program：（Diameter programming）
G02 X50.0 Z30.0 R25.0. F30； or
G02 U20.0 W-20.0 R25.0 F30； or
G02 X50.0 Z30.0 I35.0 F30（K=0）； or
G02 U20.0 W-20.0 I35.0 F30（K=0）；

Fig. 2-7 G02 arc interpolation

Note 1: One or all of Xp, Yp, Zp can be omitted. When one of them is omitted, it means the coordinate values of the starting point and the end point of the axis is consistent; when all are omitted, it means the two points are in the same position.

Note 2: When I = 0, J=0, K = 0, they can be omitted; when I, J, K and R are 0, the system executes the linear movement based on No. 3403 Bit5(CIR) or alarms.

Note 3: When I = 0, J = 0 or K = 0，and the command is executed, the tool linearly traverses to the end point.

Note 4: When I and J, J and K, I and K, are input with R, only R is valid, I, J, K are invalid.

Note 5: When the starting point and the end point are the same one, I, K are the center value, G02/G03 path is a full circle; When R is the arc radius, it means the circle is 0 degree.

Note 6: When R is the arc radius, it is more than or less than 180°, and it is more than 180° arc when R is negative; it is less than or equal to 180° when R is positive

Note 7: The alarm occurs when the radius difference between the starting point and the end point of arc exceeds the set value(except for 0) of No. 3410. When the difference does not exceed the setting value or the set value is 0, the tool firstly executes the arc interpolation along the radius value between the arc and the center, and traverse linearly to the end point; in using R programming, R should be equal to or more than the half between the starting point and the end point; when the end point is not in the arc defined by R, the user can set whether the system alarms according to No. 3403 Bit4 (RER). It is suggested that the user should use R programming.

Note 8: In G02/G03 mode, the system alarms when the other axes exceeding the current level are commanded in G02/G03

Note 9: The feedrate along the arc is related to not only F value and the override, but also the machining precision(ISB, ISC) and the machining radius. For example, when the arc radius is smaller, the machining cannot be executed at the set feedrate to get the machining precision.

Note 10: The actual cutting feedrate is limited to max. cutting feedrate MFR of No.1422

G02/G03 compound programming example:

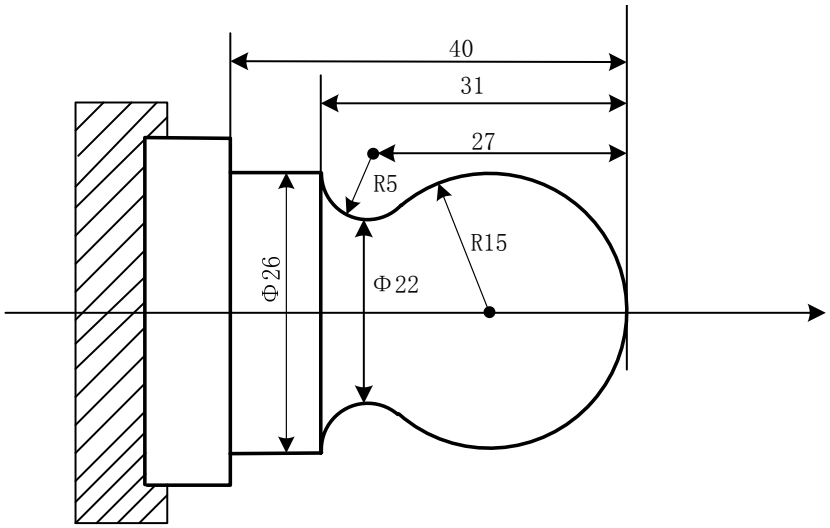


Fig. 2-8 Arc programming

```
Program：O0001
N001 G0 X40 Z5；          （Rapidly traverse）
N002 M03 S200；           （Start the spindle）
N003 G01 X0 Z0 F900；      (Approach the workpiece)
N005 G03 U24 W-24 R15；   （Cut arc R15）
N006 G02 X26 Z-31 R5；    （Cut arc R5）
N007 G01 Z-40；           (Cut φ26)
N008 X40 Z5；             (Return to starting point)
N009 M30；                （End of program）
```

2.5 Dwell G04

Command function: execute the next block after dwelling the defined time.

Command format: G04 P__ ; or
G04 X__ ; or
G04 U__ ; or
G04;

Command specification: G04 is non-modal.
The dwell time is defined by the word P__, X__ or U__.
X, U value can specify the decimal.
P value cannot have the decimal, otherwise, the system alarms.
Time of P__, X__ or U__ is shown below.

Address	P			U	X
Unit	DWT=1	0.001s		s	s
	DWT=0	ISB	0.001s		
		ISC	0.0001s		

Note: DWT is the setting value of No. 1015 Bit 7(DWT).

Value range of P__, X__ or U__ is shown below.

Address	Incremental system	Metric input	Inch input
X, U	ISB system	-99999.999～99999.999	-9999.9999～9999.9999
	ISC system	-9999.9999～9999.9999	-999.99999～999.99999
P	ISB, ISC	0~99999999	0~99999999

Note 1: The system exactly stop a block when P, X, U are not input or P, X , U specify negative values.
Note 2: X, U can command the negative value. The absolute value is taken as dwell time in G04, but the address P cannot command the negative value.

Note 3: P time unit is set by No. 1015 Bit 7(DWT).

Note 4: P, X, U are in the same block, P is valid; X, U are in the same block, the later specified command is valid.

Note 5: The dwell can be executed after the current delay time is completed in executing the feed hold in G04.

Note 6: When G04 and subprogram M98 /M99 P__ are in the same block, the number following P is the time value of G04 dwell, and is also the message of M98/M99, i.e. subprogram skip message error.

Note 7: G04 and the interpolation command in Group 1(such as G00, G01) are in the same block, G04 is valid, G0, G01 only change the modal value of G commands in Group 1.

Note 8: When No.3403 Bit 6(AD2) is 0, G04 and G commands in Group 00 are in the same block, and the later specified command is valid.

2.6 Cylindrical Interpolation 7.1

Command function: the cylindrical interpolation is defined that the movement amount of rotary axis specified by angle is converted into the movement distance of linear axis along the surface in the CNC inside, which makes the rotary axis and other axis execute the linear interpolation or circular interpolation. After interpolation, the distance is converted into the movement amount of the rotary axis, which is shown below:

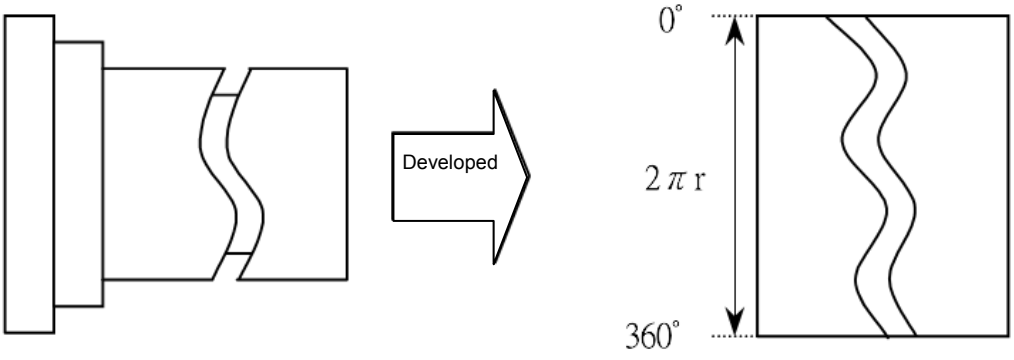


Fig. 2-9

Command format：

G07.1 {X(U) / Y(V) / Z(W) / A / B / C} r ; Activate the cylindrical interpolation code. G07.1 can be written to G107 or G7.1, but must not be with other command in a line;

……;

……;

G07.1 {X(U) / Y(V) / Z(W) / A / B / C} o ; Disable the cylindrical interpolation mode. It must not be with other command in a line;

Command explanation: G7.1 is non-modal;
r is the cylindrical radius.

Note 1: The rotary axis in the cylindrical interplation mode is specified by No. 1022, X, Y, Z or the axis parallel with it is also done. G17～G19 is specified to select the level for which the rotary axis is the specified linear axis. For example, when the rotary axis is X, G17 must specify XY level which is determined by the rotary axis and Y axis.
The rotary axis in the cylindrical interpolation mode must be set to the only one.

Before the cylindrical interpolation, the level for cylindrical interpolation must be specified firstly, otherwise, the alarm occurs; the alarm does when G17~G19 is specified to select the level when the cylindrical interpolation is being executed; G17~G19 must be specified alone with the rotary axis in the same block, otherwise, the alarm occurs.

Note 2: Even if the axis unspecified by the parameter commands the movement value in the cylindrical interpolation mode, it does not execute the cylindrical interpolation;

Note 3: The specified feedrate is the speed of the unfolded cylindrical surface in the cylindrical interpolation mode;

Note 4: One rotary axis and another linear axis can execute the circular interpolation in the cylindrical interpolation mode. But the arc radius can be specified by only R instead of I, J and K. The usage of the radius R is the same that of the circular interpolation;

The unit of the rotary axis is mm or inch instead of degree. For example, when the circular interpolation is executed between Z and C axis, No. 1022 is set to 1 (X axis) for C axis; at the

moment, the circular interpolation command is:
G18 Z__ C__;
G02(G03) Z__ C__ R__;
For C axis, when No. 1022 is set to 2, the arc command is :
G19 C__ Z__;
G02(G03) Z__ C__ R__;

Note 5: Any tool radius compensation mode being executed must be cleared before the system enters the cylindrical interpolation mode. Start and end the tool offset in the cylindrical interpolation mode; the alarm occurs when the cylindrical interpolation is enabled in the used tool radius compensation mode;

Note 6: In cylindrical interpolation mode, the movement amount of rotary axis specified by the angle is converted into the movement distance of linear axis along outerside surface, which makes rotary axis and another axis execute the linear interpolation or circular interpolation. After interpolation, the distance is converted into the angle, and the movement amount for the conversion is rounded to least input increment. So, when the diameter of the cylindrical is lesser, the actual movement amount is not equal to the specified movement amount, but the error does not acculmulate.

$$\text{Actual motion amount} = \frac{\text{MOTION\_REV}}{2\times 2\pi^2} \times \left[\text{command value} \times \frac{2\times 2\pi^2}{\text{MOTION\_REV}} \right]$$

MOTION_REV：movement amount per rotation of rotary axis（its value is set by No.1260）；
R：Radius of workpiece；
[ ]：Round to least input increment；

Note 7: In the cylindrical interpolation mode, the system alarms when the positioning operation (rapid movement command G00 and other commands to bring rapid traverse, including G28, G53, G73, G74, G76, G80～G89) cannot be specified;

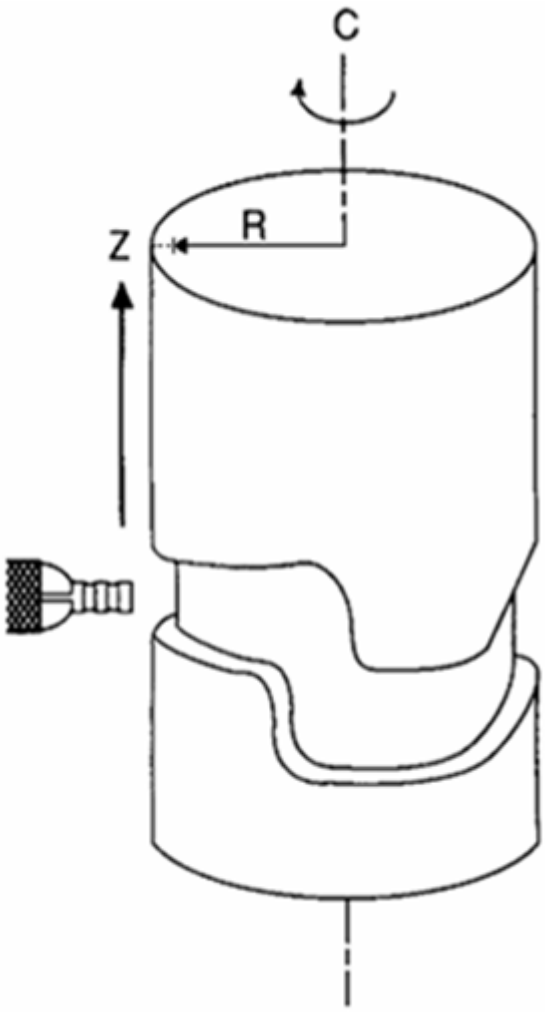
Note 8: In the cylindrical interpolation mode, the system alarms when the workpiece coordinate system（G50，G54～G59）or the local coordinate system is specified;

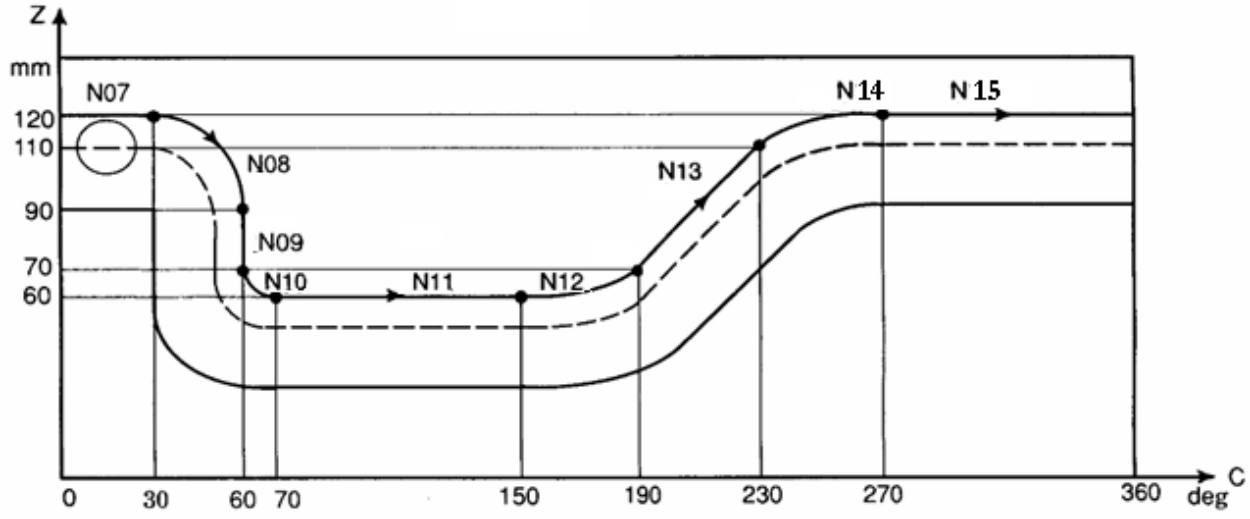
Note 9: In the cylindrical interpolation mode, the system resets to clear the cylindrical interpolation mode. It must be specified again when the syste enters the cylindrical interpolation mode again;

Note 10: The tool offset must be specified before the cylindrical interpolation mode is set, and the alarm occurs when the offset value is changed in the cylindrical interpolation mode.

Example：

```
O0001 (CYLINDRICAL INTERPOLATION);
N01 G00 Z100.0;
N02 M14; (the spindle is switched into
          position control mode)
N03 G28 H0;(zero return of C axis )
N04 G18 C0;
N05 G07.1 C67.299;
N06 G01 G42 Z120.0 F250;
N07 C30.0;
N08 G03 Z90.0 C60.0 R30.0;
N09 G01 Z70.0;
N10 G02 Z60.0 C70.0 R10.0;
N11 G01 C150.0;
N12 G02 Z70.0 C190.0 R75.0;
N13 G01 Z110.0 C230.0;
N14 G03 Z120.0 C270.0 R75.0;
N15 G01 C360.0;
N16 G40 Z100.0;
N17 G07.1 C0;
N18 M15; ( the spindle is switched into speed
          control mode)
N19 M30;
```





The above figure is the side unfolded cylindrical in the program. In the figure, when the movement amount of rotary aixs (C axis) specified by the angle is converted into the distance of linear axis of outside surface of the cylindrical, and the rotary axis and another linear axis (Z axis)

together execute interpolation, which is taken as the interpolation of Z-X level coordinate system in G18 level.

When decoding "***N07 C30.0"***, the angle movement amount of the rotary axis C si converted into the movment amount of linear axis:

$$L = \frac{\pi \times 67.299}{180} \times 30 = 35.23mm$$

Thereafter, the operation result of C's linear movement and Z's tool compensation is output to the real-time interpolation value

2.7 Polar Coordinate Interpolation G12.1, G13.1

Command function: the contour is controlled by the programming command in the rectangle coordinate system being switched into one linear motion (tool motion) and one turn motion (workpiece turn motion). The function is used to end face cutting.

Command format：G12.1； enter the polar coordinate interpolation mode, written to G112;

---------;

----------;

G13.1； cancel the polar coordinate interpolation mode, written to G113;

Command explanation: G12.1, G13.1, are specified by an single block.

After the polar coordinate mode is activated, the linear or arc interpolation in the rectangular coordinate system which consists of the linear axis and the rotary axis can be commanded.

G12.1 activates the polar coordinate interpolation mode and select a polar

coordinate interpolation level, and the polar coordinate interpolation is completed in the level.

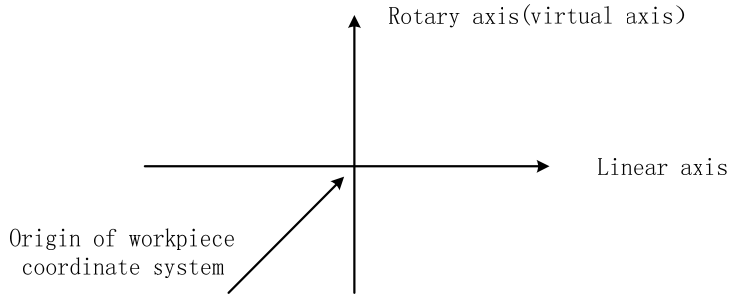


Fig. 2-10

Execution process: The polar coordinate interpolation program based on X (linear axis) and C (rotary axis).

```
O0001;
N10 T0202
        …
N100 G00 X150 C0 Z0;
N110 G12.1;
N120 G42 G01 X80 F200;
N130 C20;
N140 G03 X40 C40 R20;
N150 G01 X-40;
N160 G03 X-80 C20 R20;
N170 G01 C-20;
N180 G03 X-40 C-40 R20;
N190 G01 X40;
N200 G03 X80 C-20 R20;
N210 G01 C0;
N220 G40 X150.0;
N230 G13.1;
N240 Z100.0
        …
N500 M30
```

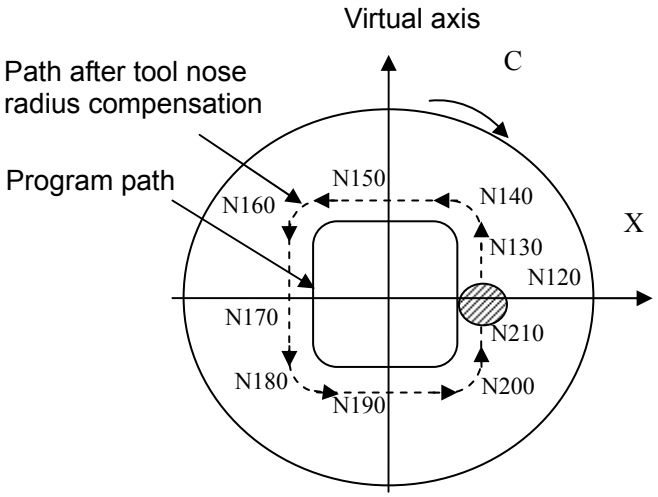


Fig.2-11

Note 1: When the system is turned on or resets, the polar coordinate interpolation is cancelled(G13.1); G12.1 and G13.1 are modal;

Note 2: The linear axis and turn axis for the polar coordinate interpolation must be set in advance in NO.5460, NO.5461; the axis undefined by the parameter does not execute the polar coordinate interpolation in spite of specifying the movement value in the polar coordinate interpolation mode;

Note 3: The used level (selected by G17, G18 or G19) before G12.1 is cancelled; after G13.1 cancels the polar coordinate interpolation, the level recovers; when the system resets, the polar coordinate interpolation is cancelled and the system uses the level selected by G17, G18 or G19;

Note 4: In the polar coordinate interpolation mode, the program commands use the rectangular coordinate command in the polar coordinate level. The linear axis in the level uses the diameter or radius programming and the turn axis uses the radius programming;

Note 5: G codes in the polar coordinate interpolation mode can be used as follows:
G01：linear interpolation;
G02, G03：arc interpolation;
G04：dwell;
G40, G41, G42：tool nose radius compensation;
G65, G66, G67：user macro program command;
G98, G99：feed/rev, feed/minute;
The system alarms when other G commands are executed in the polar coordinate interpolation mode.

Note 6: F feedrate is the tangent speed with the polar coordinate interpolation level(rectangular coordinate system) in the polar coordinate interpolation mode;

Note 7: The arc interpolation commanding the arc radius address is determined by the linear axis of the interpolation level in the polar coordinate interpolation level as follows:

Use I and J when the linear axis is X or its parallel and the turn axis uses J;
Use J and K when the linear axis is X or its parallel and the turn axis uses J;
Use K and I when the linear axis is Z or its parallel and the turn axis uses I;

Note 8: Must set a workpiece coordinate system before using G12.1, the center of the turn axis is the origin of the coordinate system. The coordinate system must not be changed in G12.1 mode.

Note 9: Cannot start or cancel the polar coordinate interpolation mode; command G12.1 or G13.1 in G40; otherwise, the system alarms;

Note 10: When the tool traverses near to the workpiece center in the polar coordinate interpolation mode, C weight of feedrate changes, which exceeds max. C cutting speed to cause the system alarms.

Note 11: The program command uses the rectangular coordinate command in the polar coordinate level. The axis address of the turn axis is taken as the one of the 2nd axis(imaginary axis) in the level. When the system executes G12.1, the tool position of the polar coordinate interpolation starts

from the angle 0. So, the spindle must be positioned before the polar coordinate interpolation is executed.

Note 12: The current position displays the actual coordinates in the polar coordinate interpolation. However, the remainder distance is displayed according to the coordinates in the polar coordinate interpolation level(rectangular coordinate level).

Note 13: Must not switch the spindle gear in the polar coordinate interpolation. The system must be in the spindle speed control mode when the gear shifting is needed.

2.8 Metric/Inch Switch G20, G21

Command function: realize the metric/inch switch of the system input mode.

Command format: G20; inch input

G21; metric input

Command explanation: G20/G21 is modal in Group 6, and can be set to the initial mode by No.0000 BIT2 (INI);

G20/G21

The units of the following value will change after they switch between the metric and the inch.

——F feedrate;

——position command;

——zero offset of workpiece;

——tool compensation value;

——scale unit of MPG;

——movement in incremental feed.

Note 1: The initial mode of G20/G21 is set by NO. 0000 BIT2 (INI) when the system is turned on.

Note 2: When G20/G21 switches the current input mode, the system must set the beginning of the program and specify in an alone block, otherwise, the system alarms.

Note 3: The tool compensation value must input the incremental unit and set it again. The tool compensation value can automatically change and cannot be set again when NO.5006 Bit0 is 1.

Note 4: It modifies NO.0000 Bit2 (INI) when the system executes G20/G21. the displayed mode also changes when NO.0000 Bit 2 (INI) is changed.

2.9 Stored Travel Check G22, G23

Command function: Create the forbidden area of stored travel limit check 2 and limit the tool traverse range in one area.

Command format： G22; stored travel 2 check is turned on

---------;

---------;

G23; stored travel 2 check is turned off

Command explanation: G22: stored travel check is turned on;

G23: stored travel check is turned off;

Positive coordinates of the stored travel area is set by No.1322;

Negative coordinates of the stored travel area is set by No. 1323;

Limit area figure: taking examples of X, Y, Z limit area are as follows. X, Y, Z are positive coordinates, I, J, K are negative.

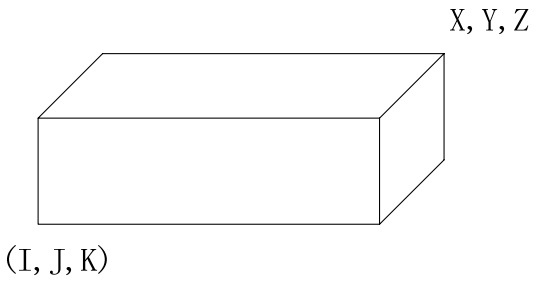


Fig. 2-12

Note 1: The initial mode of G22/G23 can be set by No. 3402 Bit 7(G23) when the system is turned on again.
Note 2: G22 stored travel check is limited to the stored travel limit check 2, and the detailed is referred to *OPERATION*;
Note 3: The data is set by the distance(min. command increment is taken as the unit) to the reference position when the parameter sets the top point of the forbidden area;
Note 4: Whether the limit range is the inner side or outer side of the area is set by No. 1300 Bit0 (OUT) , and it is the inner side when it is set to 0;
Note 5: The limit is valid after the system executes the reference position return; the system alarms when the reference position is in the limit area in G22;
Note 6: The tool reversely traverses when the travel alarm appears;
Note 7: G22/G23 is commanded in an alone block;
Note 8: The system is switched from G23 to G22 in the forbidden area, there are as follows: the system alarms in the next movement block when the forbidden area is in the inner side; alarms when the forbidden area is in the outer side;
Note 9: When the set forbidden area is set by mistaken sequence, the system executes the area check of the two points as the top points;
Note 10: When No.1310 Bit 0(OT2x) of the stored travel limit check 2 is set to 1(executing the stored travel limit 2 check), the system executes G22 and then the check; the system does not execute the check when it is G23.

2.10 Skip Interpolation G31

Command function: In the course of executing the command, when the outside skip signal (X3.5) is input, the system stops the command to execute the next block.

The function is used to the dynamic measure (such as milling machine), toolsetting measure and so on of workpiece measure.

Command format： G31 IP_ F_;

Command explanations: non-modal G command (00 group);

Its address format is same that of G01;

Cancel the tool nose radius compensation before using it;

Feedrate should not be set to too big to get the precise stop position;

The following block execution when skipping:

1. The next block of G31 is the incremental coordinate programming below.

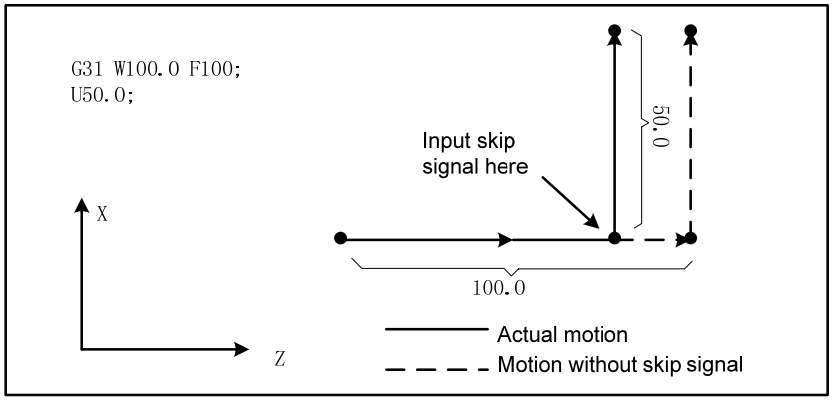


Fig. 2-13

2. The next block of G31 is the absolute coordinate programming of one axis below.

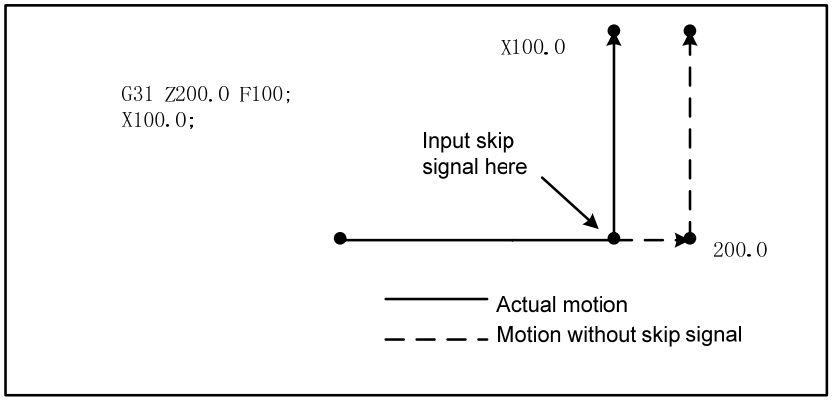


Fig. 2-14

3. The next block of G31 is the absolute coordinate programming of two axes below

Program: G31 Z200 F100

G01 X100 Z300

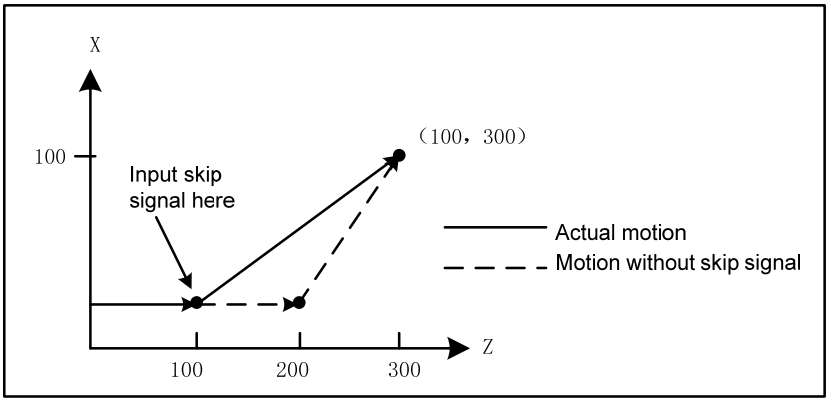


Fig. 2-15

Skip signal explanation:

SKIP signal (SKIP): X3.5

Type: input signal

Function: X3.5 ends the skip cutting. I.e. in a block containing G31, the skip signal becoming the absolute coordinate position of “1” is to be stored in the macro variable (#5061～

#5065, its last bit digit corresponds to the No. n axis of the system), at the same time, the movement in G31 block ends. No. 6200 Bit 19SK0) sets the invalid input state of the skip signal, and when it is set to 0, the input signal 1 is valid.

Operation: When the skip signal becomes “1”, CNC executes as follows: When the block is executing G31, CNC stores the current absolute coordinates for each axis. CNC stops G31 to execute the next block, the skip signal detects its state instead of its RISING EDGE. So when the skip signal is “1”, it meets the skip conditions.

Note1: When the skip signal is input, the feedrate override, the dry run, and automatic acceleration/ deceleration are invalid in the course of movement by the skip function, which is to improve the tool positioning precision.

Note 2: The skip signal is valid, the system immediately stops the feed axis (without acceleration/ deceleration execution), and G31 feedrate should be as low as possible to get the precise stop position.

2.11 Automatic Tool Offset G36, G37

Command function: When the command is executed to make the tool move to the measured position, the CNC automatically measures the difference between the current actual coordinates and the command coordinates to be the tool offset value. The function is used to the automatic toolsetting.

Command format: G36 X__;
G37 Z__;

Explanations: X absolute coordinate(only used to G36), Z absolute coordinate (only used to G37);

Non-modal G command (00 group);
Cancel the tool nose radius compensation before using it;
Only use the absolute programming;
Define the workpiece coordinate system before using the command;
Specify the tool number and tool compensation number before using the command;

Measure position arrival signal:
XAE(X3.6) ————corresponding to G36
ZAE(X3.7) ————corresponding to G37

Function: When the position measured by the program command is different from that where the tool actually reaches (i.e. at the time, the measured position arrival signal becomes “1”), the difference of the coordinates is added to the current tool compensation value to update the compensation value. When G36X_(or G37Z_) is executed, the tool firstly rapidly traverses to the position measured by the command, and decelerates and temporarily stop the position before the measured position, and then, reaches to the measured position at the speed set by No.6241 (or No.6242). When the measured position arrival signal corresponding to G command becomes the state set by No. 6240#0, and the tool is in the measured position range ±ε, the system updates the offset compensation value and ends the block. When the measured position arrival signal does not become “1”, and after the tool reaches the measured position distance ε, the

CNC alarms, ends the block and does not update the offset compensation value.

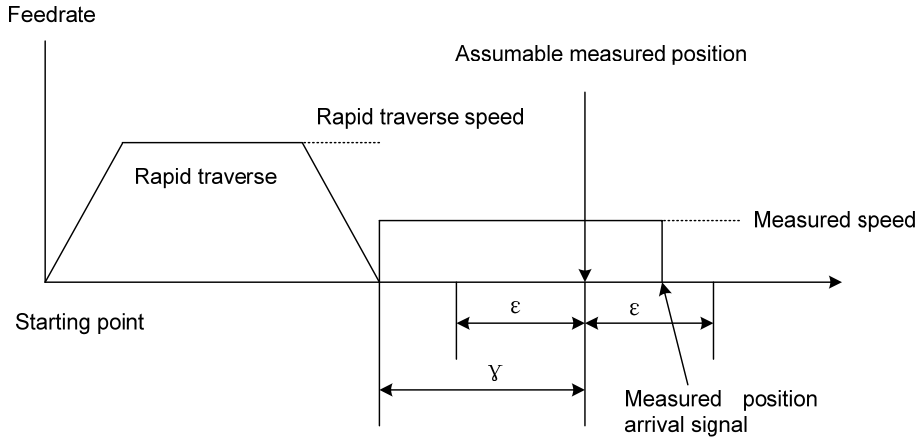
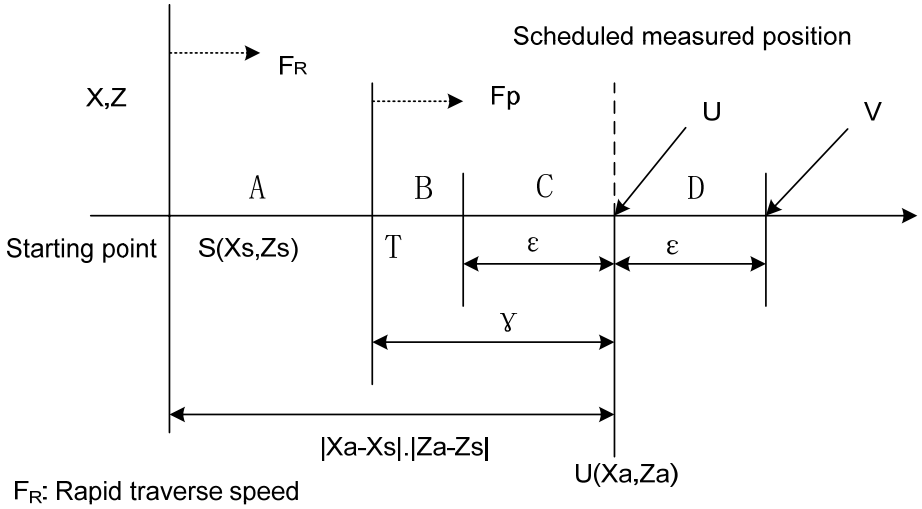


Fig. 2-16

G36, G37 automatic tool offset command use

From the initial position to the measured position specified by Xa or Za in G36 or G37, the tool rapidly traverses to A zone and stops at T point (Xa–γx or Za–γz) , and then traverses to B, C and D at the feedrate set by No.6241(or No.6242). The system alarms when the tool traverses in B zone and the measured point arrival signal of the end point is set to. The system alarms when the tool stops at V point. Parameter No. 6241, No. 6242, No.6254, No.6255 are set by the radius value.



F_R: Rapid traverse speed
Fp: Feedrate set by No. 6241
γ: Parameter (No.6251, No.6252)
ε : Parameter (No.6254, No.6255)

Fig. 2-17

Example:

```
G50 X760 Z1100;   create the workpiece coordinate system
T0101;            define No. 1 tool and execute its tool compensation
G36 X200;         traverse to X toolsetting point ( X toolsetting point coordinate:
                  200)
T0101;            execute X tool compensation again
G00 X204;         retract a little
G37 Z800;         traverse to Z toolsetting point ( Z toolsetting point coordinate:
                  800)
T0101;            execute Z tool compensation again and the toolsetting is completed
```

M30;

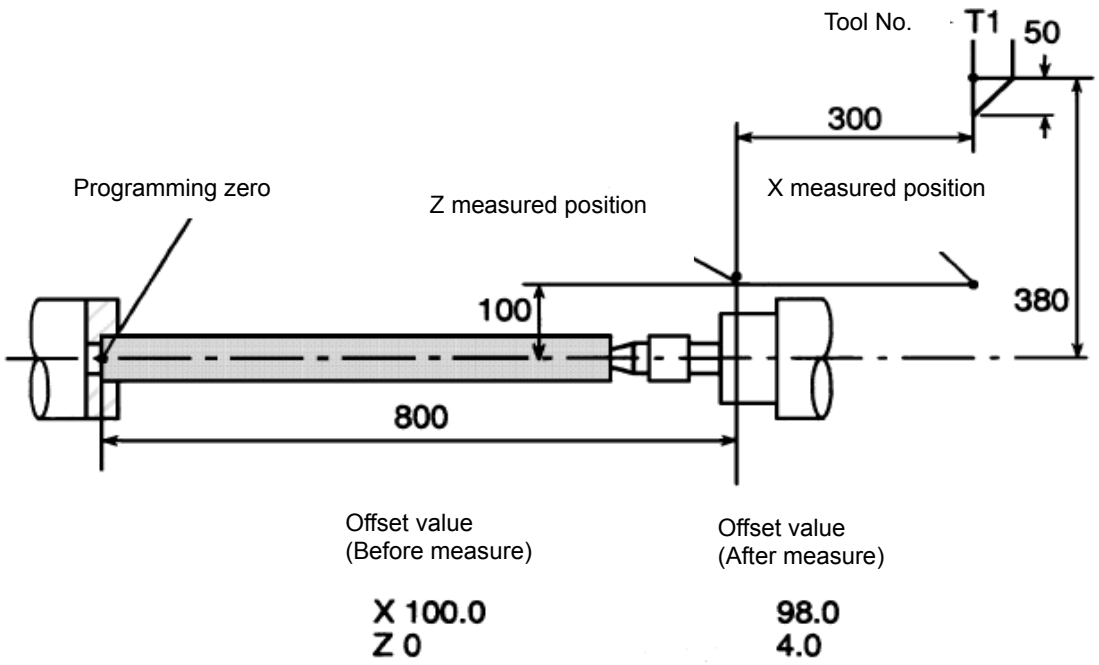


Fig. 2-18

2.12 Reference Position Function

2.12.1 Reference position return G28

Command function: move from the starting point at the rapid traverse speed to the middle position specified by IP_ and then return to the reference position.

Command format: G28 IP__ ;

Command explanation: G28 is non-modal.

IP_: it is the middle point coordinates, is specified by the absolute value and incremental value. Omit one or all command address for each axis, omitting some axis means the axis does not return to the reference position, omitting all means the middle point is the tool starting point in the current workpiece coordinate system, and the tool does not return to the reference position and keeps stopping.

Command execution process: (as Fig. 2-18):

（1）Rapidly position from the current position to the middle position of the command axis(A→B);

（2）Rapidly position from the middle point to the reference position (B→R);

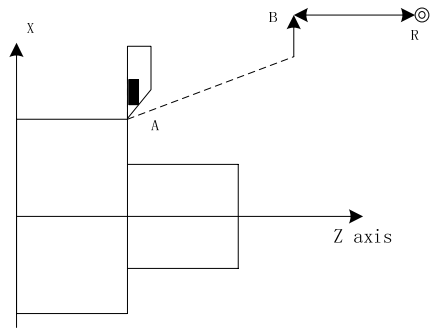


Fig. 2-19

Note 1: After the system is turned on, it does not execute the manual reference position return; when the system executes G28 reference position return, it judges it alarms or executes like the manual reference position return according to No. 1002 Bit 3(AZR) to use the deceleration block to execute the reference position return. But, when the reference position setting function without the block(No.1002 Bit1 (DLZ)) is set to 1 or NO.1005 Bit 1(DLZx) is set to 1, it is unrelated to AZR setting, the system alarms when the system executes G28 before the reference position is created.

Note 2: Each axis separately moves at the rapid traverse speed from the starting point through the middle point to the reference position, i.e. G00 mode.

Note 3: G28 or G30 in the tool radius compensation mode automatically cancels the tool radius compensation, and automatically recovers it in the next movement command.

Note 4: Generally, G28 is specified in an alone line; when the system specifies simultaneously the same parameter address word of G00 or G01, IP_ is specified to G28 parameter, G00 or G01 only change the modal value of the corresponding G groups and does not execute the motion.

2.12.2 2nd, 3rd, 4th reference position return G30

Command function: move at the rapidly traverse speed to the middle point specified IP_ and then to the 2nd, 3rd and 4th reference position.

Command format： G30 P2 IP__ ； return to the 2nd reference position
G30 P3 IP__ ； return to the 3rd reference position
G30 P4 IP__ ； return to the 4th reference position

Command explanation: G30 is non-modal;

IP_: it is the middle point coordinates, is specified by the absolute value and incremental value. Omit one or all command address for each axis, omitting some axis means the axis does not return to the reference position, omitting all means the middle point is the tool starting point in the current workpiece coordinate system, and the tool does not return to the reference position and keeps stopping.

Command execution process (as Fig.2-18):

（1）Rapidly position from the current position to the middle position of the command axis(A→B);

（2）Rapidly position from the middle point to the reference position (B→R);

Note 1: Reference position position is set in NO.1241～NO.1243;

Note 2: After the system is turned on, it executes the reference position return once before executing G30; do not execute the reference position return firstly before executing G30 after the system with the absolute encoder is turned on;

Note 3: When P is omitted, the system executes it as P2 and returns to the 2nd reference position;

Note 4: The middle point will move to the new workpiece coordinate system when the workpiece

coordinate system is changed;

Note 5: Each axis separately moves at the rapid traverse speed from the starting point through the middle point to the reference position, i.e. G00 mode.

2.13 Related Function of Coordinate System

The tool position is expressed with the coordinate value of the coordinate system, the coordinate value is specified by the programmed axis. GSK988T system has three kinds of coordinate system:

1. machine coordinate system
2. workpiece coordinate system
3. local coordinate system

Fig.2-19 describes the relationship of the three coordinate systems:

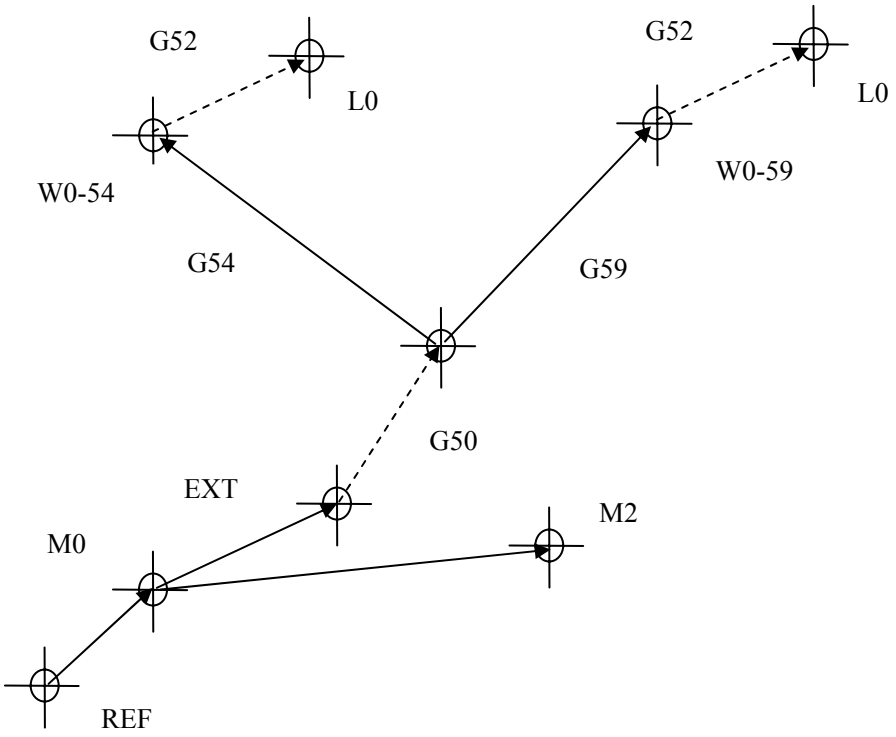


Fig. 2-12

REF	Reference position.
M0	Origin of machine coordinate system is a fixed point on the machine, No. 1240 value confirms the relative position of the reference position and the machine origin.
M2	The 2nd reference position, No.1214 set the 2nd reference position position in the machine coordinate system.
EXT	The outer origin offset can be set by No. 1220 or in the coordinate setting window.
G50	The offset set by G50 is 0 when the system is turned on.
G54, 59	The offset of the workpiece coordinate system is set by No. 1221, No. 1226, and is also set in the coordinate window.
W0-54, W0-59	Origin of the workpice coordinate system.
G52	The offset of the local coordinate system is 0 when the system is switched on. All workpiece coordinate systems share, i.e. the local coordinate system offset set in one workpiece coordinate system can exist in other workpiece coordinate system.
L0	Origin of the local coordinate system.

Note: The system has created the above coordinate system after the 1st reference position return is executed. The coordinate system is created after the system is turned on with the absolute position encoder.

2.13.1 Selecting machine coordinate system position G53

A particular on the machine as the machining reference is called as the machine zero which is taken as the origin of the coordinate system is called as the machine coordinate system. After the system is turned on, executing the manual reference position return sets the machine coordinate system which keeps till the system is turned off.

Command format: G53 IP__ ;

Command function: when the position of the machine coordinate system is commanded, the tool moves the position at the rapid traverse speed. Omitting one axis means the axis does not move; when the system only specifies G53 without specifying the positions of any axes, the system does not execute the motion.

Command explanation: G53 is non-modal;

IP_: the absolute coordinate value for each axis in the machine coordinate system must be specified by the absolute value.

As the following figure: the specified axis rapidly moves from A (20, 20) in the current workpiece coordinate system to B (-8, -10) in the machine coordinate system.

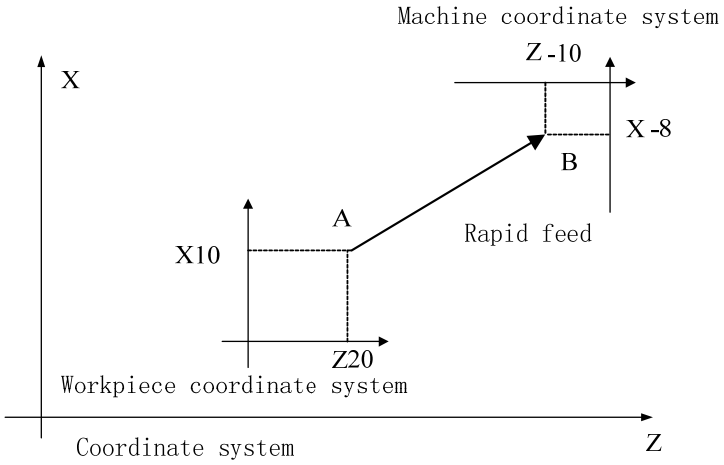


Fig.2-21

Note 1: G53 is non-modal, and is valid in other blocks;
Note 2:G53 specifies the absolute position value in the machine coordinate system. The axis command is ignored when some axis uses the incremental value command;
Note 3: When G53 is commanded, the system cancels the tool nose radius compensation;
Note 4: After the system is turned on, the system performs the manual reference position return or G28 automatic reference position return, and automatically creates the origin position of the machine coordinate system according to the value set by No. 1240;
Note 5:The machine coordinate system must be set before the system commands G53. So, the system must execute the manual reference position return or G28 automatic reference position return after it is turned on; the operation is not operated when the system uses the absolute position encoder;
Note 6: The system executes G53 and G00, G01 in Group 01 in the same block, G00 or G01 only modifies G modal value in Group 01.

2.13.2 Workpiece coordinate system setting G50

The coordinate system used to machining the workpiece is called as the workpiece coordinate system.

The workpiece coordinate system can be set in advance. The set workpiece can change its origin position to set again the position of workpice coordinate system in the machine coordinate system.

Command format: G50 IP__ ;

Command function: The absolute coordinate of the current position can be set by setting the absolute coordinate of current position to create the workpiece coordinate system (called as the floating coordinate system). After the workpiece coordinate system is created, the absolute coordinate programming inputs the coordinate value in the coordinate system till the new workpiece coordinate system in G50 is created.

Command explanation: G50 is non-modal G;

IP_: When the system uses the absolute command, it specifies the new absolute coordinate position of the current point in the coordinate system; when the system uses the incremental command, after its executes G50, the absolute coordinate value of the current point is equal to the sum between the absolute coordinate value before execution and the coordinate incremental value.

Note 1: After G50 changes the workpiece coordinate system, other workpiece coordinate system also performs the same offset;

Note 2: In G50, the system can omit one or all command addresses for each axis, the current coordinate value is not input when the command value for each axis is not input. When the axis command address is omitted, the coordinate axis which is not input keeps its pervious coordinate value;

Note 3: When G50 and G command (G00, G01) are in the same block, the system only modifies the modal value of Group 1, and the coordinate value in the block is specified by G50;

Note 4: When the system does not set G50 offset value, it can set No. 1202 Bit(G50) to forbid G50;

Note 5: After G50 sets the coordinate system, the system must be turned off and then on, the coordinate values set by G50 remain unchanged before power off.

Note 6: In NC program, when LGT is set the coordinate offset mode to execute the tool offset, and the system executes T function does not execute the absolute value command, the coordinate system is set by G50, the absolute coordinate value displayed by G50 is the one that the coordinate value set by G50 adding the tool compensation value which is not executed. The difference between the relative coordinates and the machine coordinates is （-80，10）when the system executes N4, the difference value is caused because X100Z10 setting G50X20Z20 to create the workpiece coordinate system offset, i.e. the user does not think over the tool offset influence when G50 is set in NC program.

Program	Absolute coordinate	Relative coordinate	Machine coordinate
N1 T0100 G00 X100 Z10	X：100 Z：10	X：100 Z：10	X：100 Z：10
N2 T0101（No.01 tool compensation value X12 Z23）	X：88 Z：-13	X：100 Z：10	X：100 Z：10
N3 G50 X20 Z20	X：8 Z：-3	X：20 Z：20	X：100 Z：10
N4 G00 X10 Z10	X：10 Z：10	X：22 Z：33	X：102 Z：23

2.13.3 Workpiece coordinate system selection command G54～G59

Command function: One of G54～G59 is specified, one of workpice coordinate system 1～6 can be selected. After the workpiece coordinate system is specified, the specified point in the block is in the specified workpiece till a new workpiece coordinate system is created as Fig. 2-21. The tool positions X60.0, Z20.0 in the workpiece coordinate system 3.

G56 G00 X60.0 Z20.0

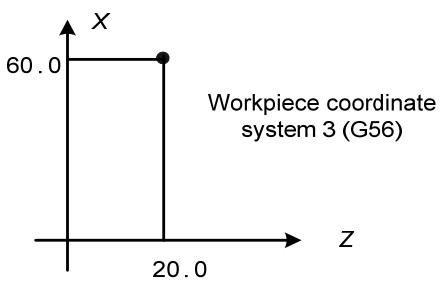


Fig. 2-22

Command format：G54 workpiece coordinate system 1；

G55 workpiece coordinate system 2；
G56 workpiece coordinate system 3；
G57 workpiece coordinate system 4；
G58 workpiece coordinate system 5；
G59 workpiece coordinate system 6；

Command explanation: G54～G59 are modal.

Note 1: The workpiece is created after the system is turned on and executes the reference position return. When the system is turned on, it automatically selects G54 as the current workpiece coordinate system;

Note 2: G54-G59 describing the 6 workpiece coordinate systems can change their positions by the external workpiece zero offset value or workpiece zero offset value, and their relationship is as Fig. 2-22;

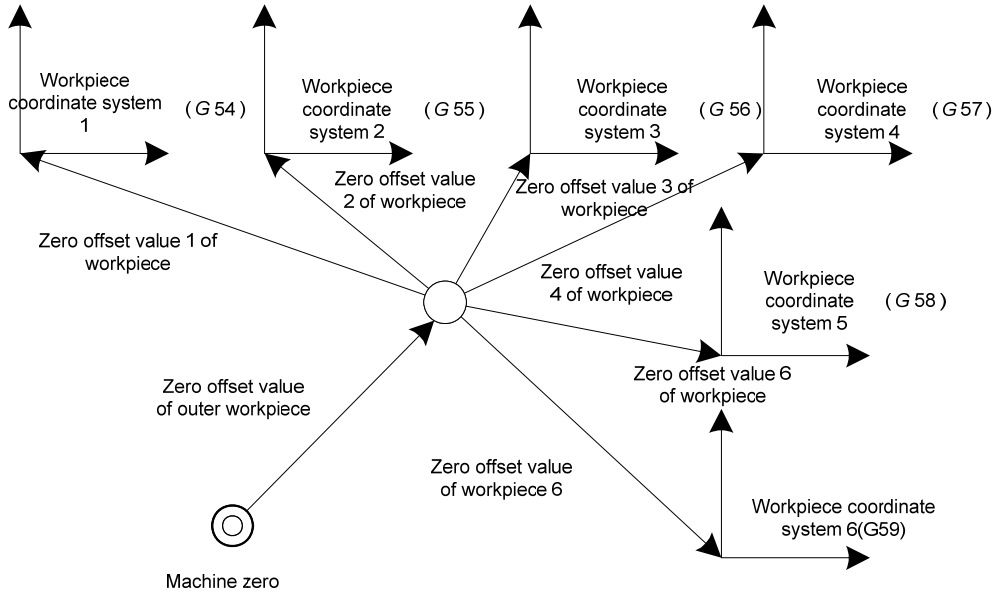

Workpiece coordinate system 1
(G 54)
Workpiece coordinate system 2
(G 55)
Workpiece coordinate system 3
(G 56)
Workpiece coordinate system 4
(G 57)
Zero offset value 2 of workpiece
Zero offset value 3 of workpiece
Zero offset value 1 of workpiece
Zero offset value 4 of workpiece
Workpiece coordinate system 5
(G 58)
Zero offset value 6 of workpiece
Zero offset value of outer workpiece
Zero offset value of workpiece 6
Workpiece coordinate system 6(G59)
Machine zero

Fig. 2-23

Note 3: Use the following method to change:

1）MDI input changes the workpiece coordinate system zero;

2）Use G50 to move the workpiece coordinate system;

Specifying G50 IP_ makes the workpiece coordinate system（G54～G59）to set a new workpiece coordinate system where the current tool position is consistent with the specified coordinates. When G50 specifies the relative value, the value adding the previous tool position coordinate value creates a new coordinate system, but the tool position does not change but the coordinate system executes the offset as Fig. 2-23:

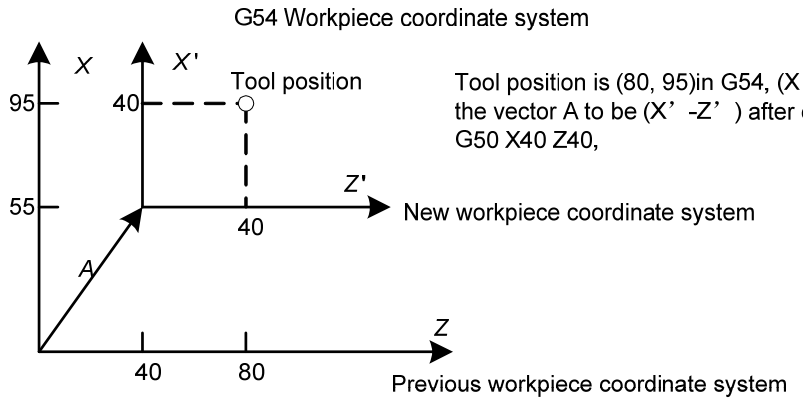
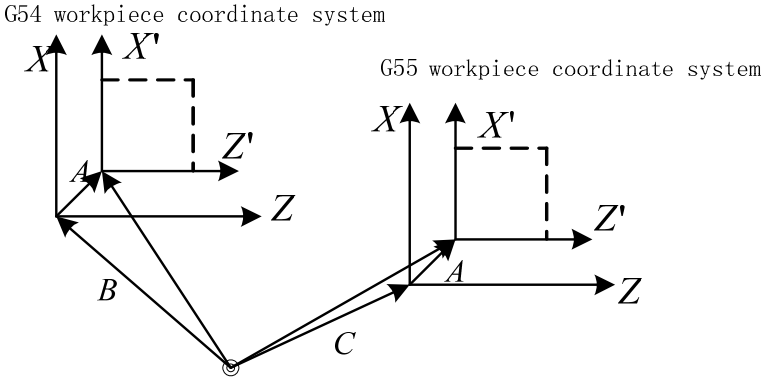


Fig. 2-24

Note 4: The coordinate offset value created by G50 adds to the one of all workpiece zero to make ensure that all workpiece coordinate systems offset the same value as Fig. 3-21:



$X'-Z'$, New workpiece coordinate system

$X-Z$, Previous workpiece coordinate system

A: $G50$ Offset value

B: Zero offset value of workpiece in G54

C: Zero offset value of workpiece in G55

Fig. 2-25

Note 5: The workpiece zero offset value of G54～G59 workpiece coordinate system can be set in the parameters and input in the coordinate setting window;

Note 6: When the system is turned on, it defaults G54 as the current workpiece coordinate system; after the system executes the reference position return, it creates the coordinate system, uses G55～G59 to switch to other workpiece coordinate system; when the system resets, No.1201 Bit 7(WZR) determines whether the system returns to G54 workpiece coordinate system; when No. 3402 Bit 6(CLR) is set to 1, the modal returns to G54.

2.13.4 Local coordinate system setting G52

To be convenient to programming, the sub-coordinate system to set the workpiece coordinate system is called the local coordinate system.

Command format: G52 IP__; set the local coordinate system

……

G52 IP0; cancel the local coordinate system (IP0 means the absolute value for each axis adds one zero)

Command function: commanding G52 in the program can set the local coordinate system in the workpiece coordinate system G54~G59. The origin of the local coordinate system can set in the position specified by IP_ in the workpiece coordinate system. The corresponding relationship is as Fig. 2-26.

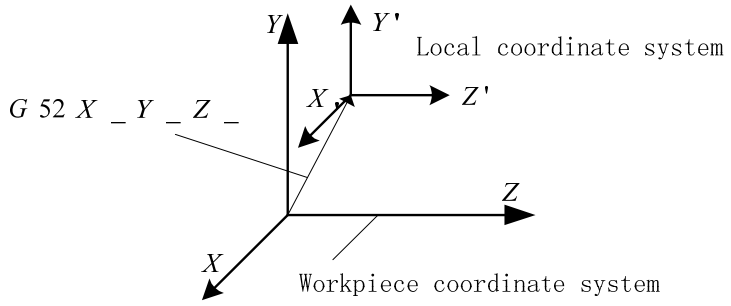


Fig. 2-26

Command explanation: G52 is non-modal;

IP_: when IP_ is absolute command, the system specifies the absolute coordinate value of origin of local coordinate system in the workpiece coordinate system; when IP_ is the incremental command, the system specifies the relative coordinate value of the origin of the local coordinate system related to the one of the workpiece coordinate system;

Once the local coordinate system is created, its coordinates are used to the axis motion command. Using G52 to command the zero of the new local coordinate system(workpiece coordinate system) can change the position of the local coordinate system;

Making the zero of the local coordinate system coincide with the one of the workpiece coordinate system can cancel the local coordinate system and returns to the workpiece coordinate system, i.e. command G52 X0 Z0.

Note 1: The local coordinate system setting does not change the workpiece coordinate system and the machine coordinate system.

Note 2: Commanding G52 can temporarily cancel the offset in the tool nose radius compensation.

Note 3: In local coordinate system, when G50 sets the workpiece coordinate system and the system has not specified the coordinate values to all axes in the local coordinate system, the axis which is not specified in G50 in the local coordinate system still keeps, the local coordinate system corresponding to G50 axis is cancelled; For example:

……

G52 X50 Z50;

……

G50 X100; at the moment, Z coordinate value is not change, the local coordinate system corresponding to X is cancelled

……

Note 4: When the system selects the workpice coordinate system command （G54～G59）to change the workpiece coordinate system in the local coordinate system, the local coordinate system also moves to the new workpiece coordinate system.

Note 5: Whether the local coordinate system in reset is cancelled is determined by No.1202 Bit 3(RLC) , the local coordinate system is cancelled in reset when the parameter is set to 1.

Note 6: Whether the local coordinate system in manual reference position return is cancelled is determined by No.1201Bit 2 (ZCL), the local coordinate system is cancelled in manual

reference position return when the parameter is set to 1.

2.13.5 Level selection command G17～G19

Command function: The level selection command is used to the arc interpolation and the tool nose radius compensation selection level. Once the system has selected the level, it can execute the arc interpolation and tool nose radius compensation on the level.

Command format: G17 selects XpYp level;
G18 selects ZpXp level;
G19 selects YpZp level;

Command explanation: G17, G18, G19 are modal G commands.
Xp: X or its parallel axis
Yp: Y or its parallel axis
Zp: Z or its parallel axis

Note 1: Xp, Yp, Zp are determined by the axis addresses of G17, G18, G19 in the block; when the axis addresses are omitted, the system defaults the omitted are the addresses of the basic axis; the level keeps when the system does not command G17, G18, G19 blocks.

Note 2: The parameter sets each axis to have three basic axes (X, Y, Z) or the parallel axis.

Note 3: The level remains unchanged in the G17, G18, G19 not be specified.

Note 4: When the system is turned on, its initialization is defaulted to G18 state, i.e. ZX level;

Note 5: When the system repetitively specifies G17～G19 in the same block, and No.3403 Bit 6(AD2) is 0, the last G17～G19 word is valid, the system alarms when the parameter is set to 1;

Note 6: The multi-compound cycle command（G70～G76） and the fixed cycle command（G90, G92, G94）are used to ZX basic axis level; when their functions are specified in other levels, the system alarms;

Note 7: The motion command is not related to the level selection, besides the arc interpolation and tool nose radius compensation command, when the system commands the axis beyond the levels, it does not alarm and the axis can move; when the system selects the axis motion beyond the level in the arc interpolation command, the system alarms. For example:

```
……;
G17;
G01 X100 Y50 Z20 F100; the system does not alarm, Z moves
……;
G02 X20 Z50 R100; the system alarms
……;
```

Example: the level selection: when X and A are parallel axis:

```
G17 X_ Y_ ； select XY level
G17 A_ Y_ ； select AY level
G18 X_ Z_ ； select ZX level
G17；          select XY level
G17 A_         select AY level
G18 Y_         select ZX level, Y motion is not relative the level
```

2.13.6 Exact stop mode G61/cutting mode G64

G61 function: After programmed axis of the block must exactly stop at the end pont of the block, the next block is executed.

G64 function: When the programmed axis of each block following G64 starts to develerate (it has not reached the programmed end point), the system starts to execute

the next block, the programmed contour in G64 is different from the actual, and the difference condition is determined by F value and the angle between two paths, the more the different is, the more F value is.

Command format: G61; （exact stop mode）

G64; （cutting mode）

Command explanations:

1. A block including G61 eactaly stops the end point of the program before the system executes the next block, which is used to process sharpt edges and corners. G61 is modal and valid till G64 is commanded. The programmed contour is the same that of the actual.
2. G64 is modal, valid and default before G61 is commanded. G64 path is different from that of G61 as Fig. 2-27;
3. G61, G64 belong to Group 15, and their relations with other G groups are referred to Group 5.
4. When G01 is executed, it is in the exact stop in cutting mode because it is non cutting command.
5. When G61/G64 is specified, it is value in the next commanded block.

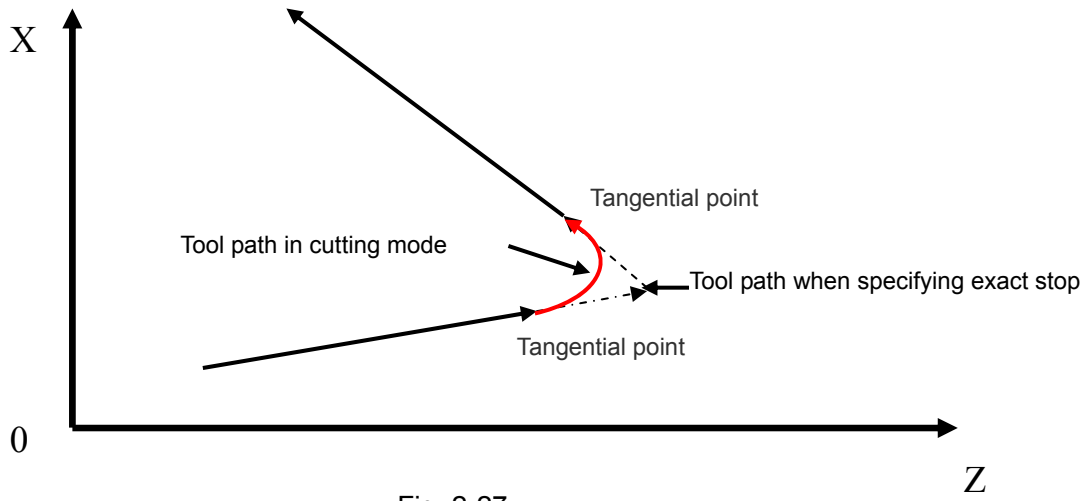


Fig. 2-27

Note: The system defaults G64 cutting mode.

2.14 Fixed Cycle Command

To simplify programming, the system defines G command of single machining cycle with one block to complete the rapid traverse to position, linear/thread cutting and rapid traverse to return to the starting point:

G90: axial cutting cycle; G92: thread cutting cycle; G94: radial cutting cycle;

G92 thread cutting fixed cycle command is described in **Thread Function**.

2.14.1 Axial cutting cycle G90

Command function: From starting point, the cutting cycle of cylindrical surface or taper

surface is completed by radial feeding(X) and axial (Z or X and Z) cutting.

Command format：G90 X（U）__ Z（W）__ F__；　（cylinder cutting）

G90 X（U）__ Z（W）__ R__ F__；（taper cutting）

Command specifications:

G90 is modal;

X_,Z_	Coordinates of longitudinal cutting (C point in the figure below)
U_,W_	Movement to end point (C point in the figure below) of longitudinal cutting
F_	Cutting feedrate
R_	Taper（radius value, with direction, range referred to the table below

Address	Incremental system	metric（mm）input	Inch (inch) input
R	ISB system	-99999.999～99999.999mm	-9999.9999～9999.9999 inch
	ISC system	-9999.9999～9999.9999 mm	-999.99999～999.99999 inch

Cycle process:

① X rapidly traverses from starting point to cutting starting point;

② Cutting feed (linear interpolation) from the cutting starting point to cutting end point;

③ X executes the tool retraction at feedrate (opposite direction to the above-mentioned ①), and return to the position which the absolute coordinates and the starting point are the same;

④ Z rapidly traverses to return to the starting point and the cycle is completed.

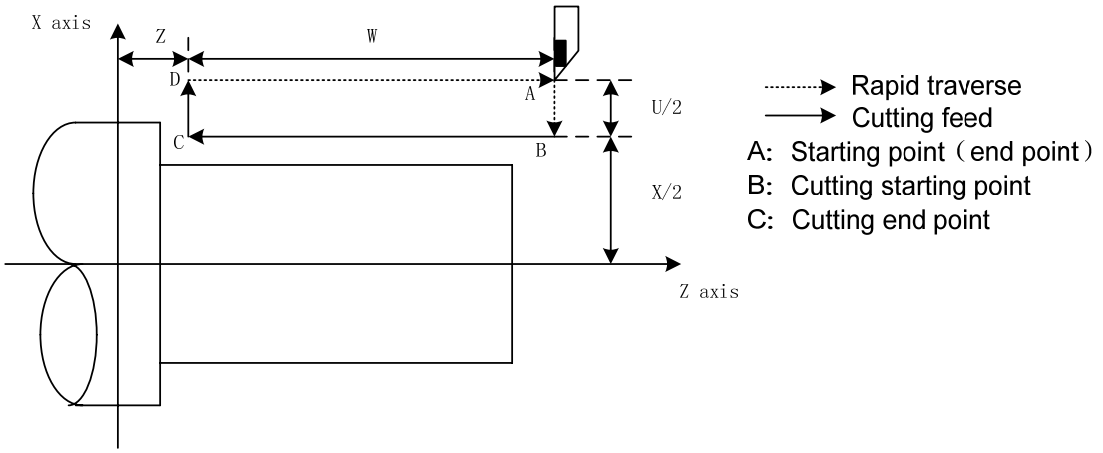


Fig.2-28

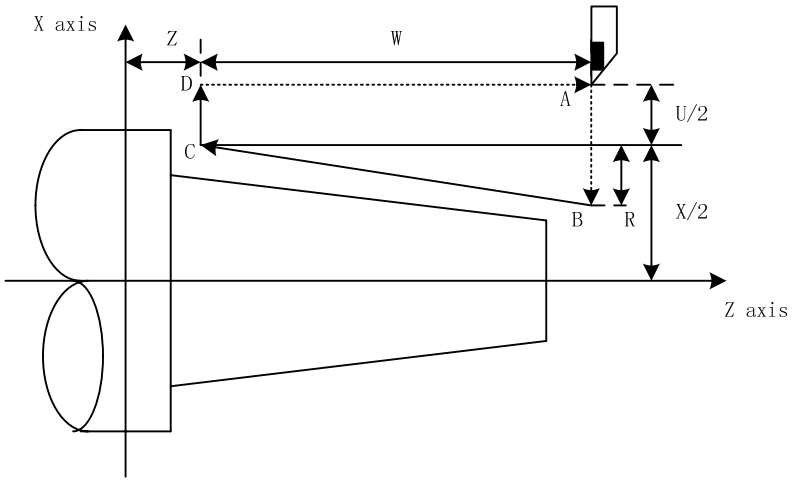
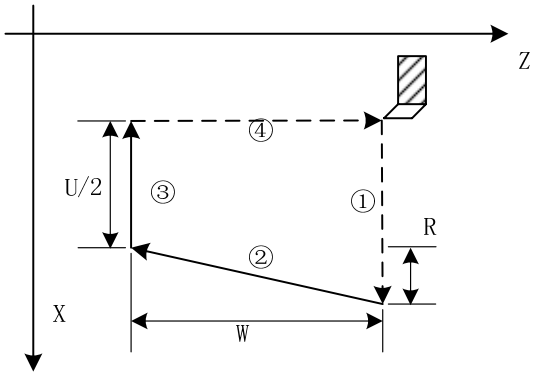


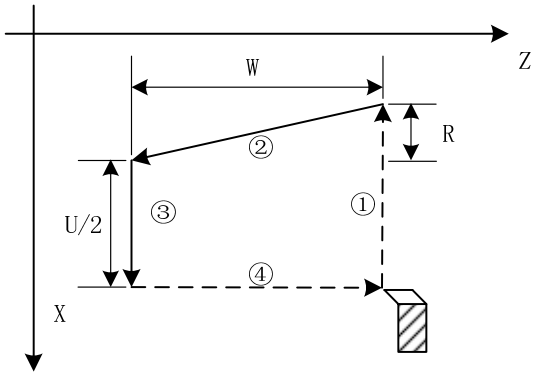
Fig. 2-29

Cutting path: Relative position between cutting end point and starting point with U, W, R, and tool path of U, W, R with different sign symbols are as Fig. 2-28:

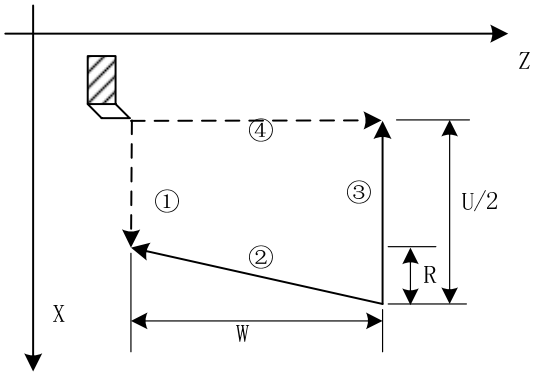
1） U>0，W<0，R>0



2） U<0，W<0，R<0



3） U>0，W>0，R<0 ,|R|≤|U/2|



4） U<0，W>0，R>0,|R|≤|U/2|

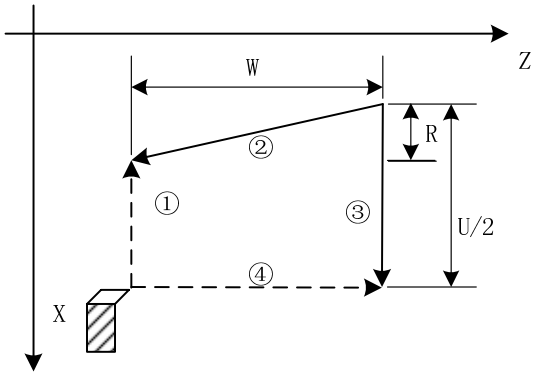


Fig.2-30

Example: Fig. 2-29, rod Φ125×110

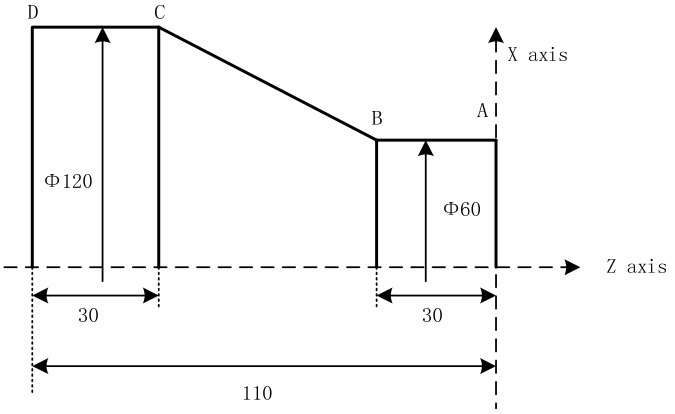


Fig.2-31

Program：

```
Program : O0002;
M3 S300 G0 X130 Z3;
G90 X120 Z-110 F200;                    (A→D, cut Φ120)
X110 Z-30;
X100;
X90;                                    （A→B，6 times cutting cycle Φ60, increment of 10mm）
X80;
X70;
X60;
G0 X120 Z-30;
G90 X120 Z-44 R-7.5 F150;
Z-56 R-15                               （B→C，4 times taper cutting）
Z-68 R-22.5
Z-80 R-30
M30;
```

2.14.2 Radial cutting cycle G94

Command function: From starting point, the cutting cycle of cylindrical surface or taper surface is completed by radial feeding(X) and axial (Z or X and Z) cutting.

Command format: G94 X(U) __ Z(W) __ F__; (face cutting)

G94 X(U) __ Z(W) __ R__ F__; (taper face cutting)

Command specifications: G94 is modal;

X_,Z_	Coordinate of cutting end point(C point in the figure below)in the direction of the bottom side
U_,W_	Movement to cutting end point (C point in the figure below)in the direction of bottom side
F_	Cutting feedrate
R_	Taper Taper（radius value, with direction, range referred to the table below

Address	Incremental system	Metric（mm）input	Inch (inch) input
R	ISB system	-99999.999～99999.999mm	-9999.9999～9999.9999 inch
	ISC system	-9999.9999～9999.9999 mm	-999.99999～999.99999 inch

Cycle process:

① Z rapidly traverses from starting point to cutting starting point;

② Cutting feed (linear interpolation) from the cutting starting point to cutting end point;

③ Z executes the tool retraction at the cutting feedrate (opposite direction to the above-mentioned ①), and returns to the position which the absolute coordinates and the starting point are the same;

④ The tool rapidly traverses to return to the starting point and the cycle is completed.

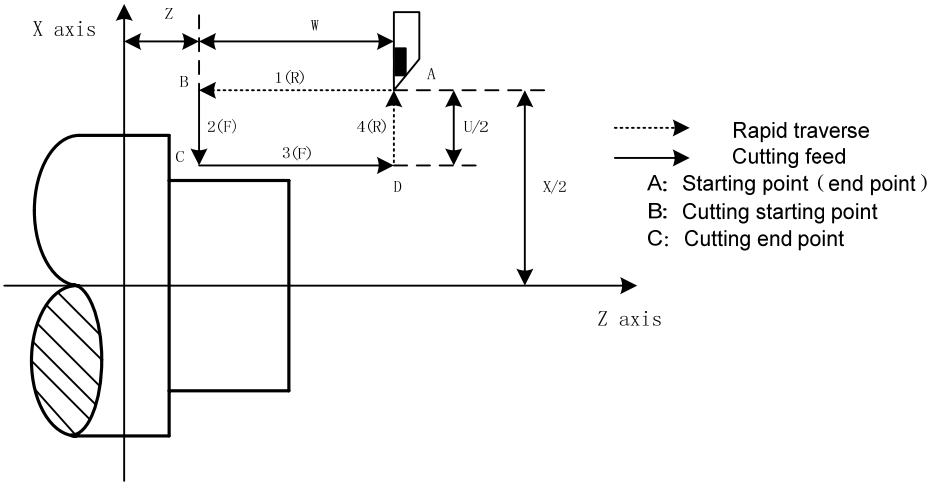


Fig. 2-32

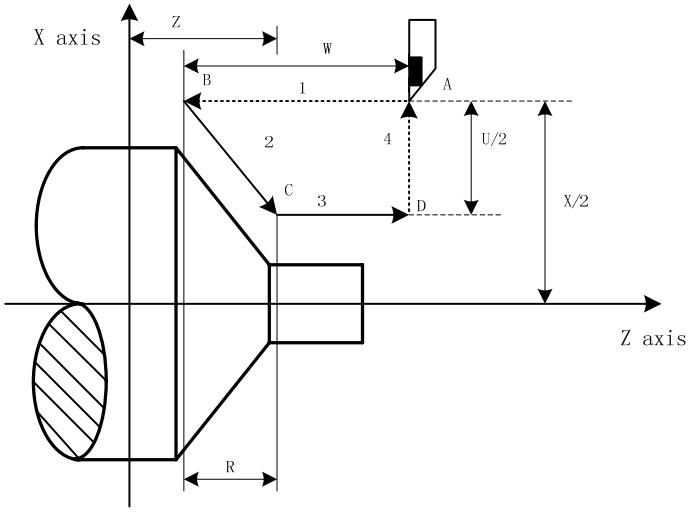
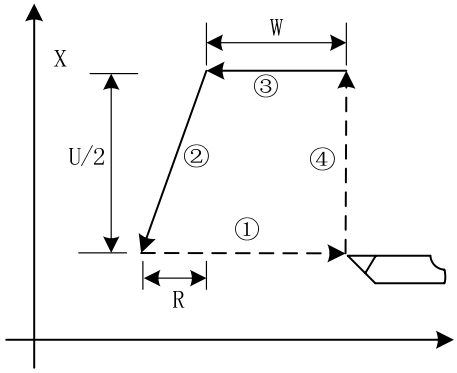


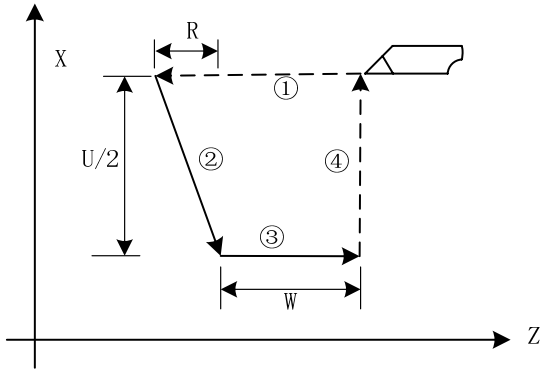
Fig.2-33

Cutting path: Relative position between cutting end point and starting point with U, W is as Fig.2-32:

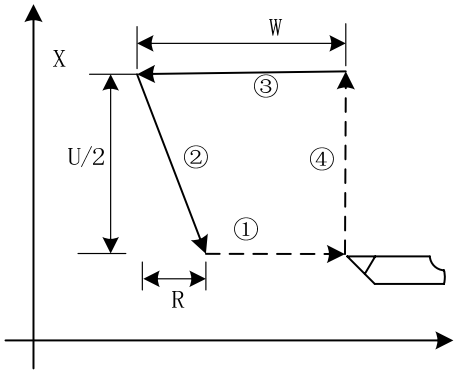
1） U>0 W<0 R<0



2） U<0 W<0 R<0



（3）U>0 W>0 R<0 （|R|≤|W|）



4）U<0 W>0 R<0 （|R|≤|W|）

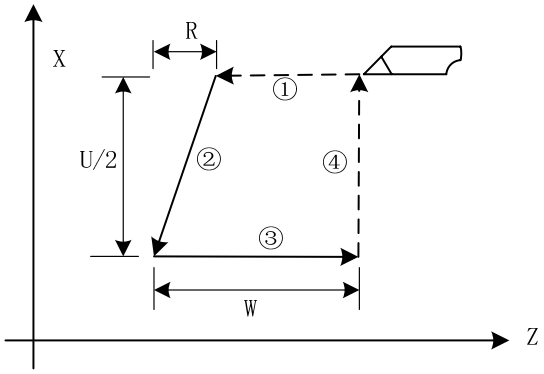
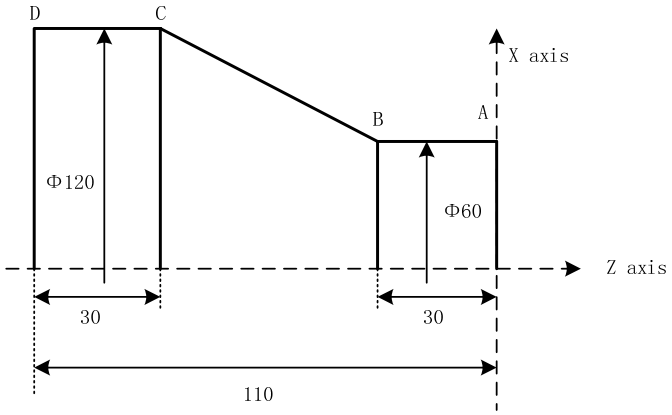


Fig. 2-34

Example: Fig. 2-35, rob Φ125×112



Program：

```
G00 X130 Z5 M3 S1；
G94 X0 Z0 F200
X120 Z-110 F300；
G00 X120 Z0
```

End face cutting

（cut outer Φ120）

```
G94 X108 Z-30 R-10
X96 R-20
X84 R-30
X72 R-40
X60 R-50;
M30;
```
（C→B→A，cut Φ60）

Note 1:These fixed cycle commands are used to ZX level. The system alarms when other axis motion in the block of the fixed cycle command is commanded;

Note 2: After X(U) , Z(W) , R are executed in the canned cycle command, their command values are value if X(U) , Z(W) ,R are not redefined by executing a new canned cycle commands. The command values of X(U) ,Z(W) ,R are cleared if non-modal G command(00 Group) except for G04 or G00, G01, G02, G03, G32 is executed;

Note 3: In MDI mode, the previous canned cycle can be executed by pressing the cycle start key after the canned cycle is completed;

Note 4: One cycle cannot be executed repetitively in G90～G94 when the next block of G90～G94 is M, S, T command; the previous cycle is executed repetitively in G90～G94 when the next block is ended(EOB;).

Example …

```
N010 G90 X20.0 Z10.0 F400;
N011 ;                    （execute G90 one time again）
…
```

Note 5: Pause or single block is executed in G90, G94, the single block stops after the tool moves end point of current path.

2.15 Multiple Cycle Commands

GSK988T multiple cycle commands include axial roughing cycle G71, radial roughing cycle G72, closed cutting cycle G73, finishing cycle G70, axial grooving multiple cycle G74, axial grooving multiple cycle G75 and multiple thread cutting cycle G76. When the system executes these commands, it automatically counts the cutting times and the cutting path according to the programmed path, travels of tool infeed and tool retraction, executes multiple machining cycle(tool infeed →cutting→retract tool→tool infeed), automatically completes the roughing, finishing workpiece and the starting point and the end point of command are the same one.

G76 multiple thread cutting cycle command is described in **Thread Function**.

2.15.1 Axial Roughing Cycle G71

Command function: G71 is divided into three parts:

⑴: 1st blocks for defining the travels of tool infeed and retract tool, the cutting feedrate, the spindle speed and the tool function when roughing;

⑵: 2nd blocks for defining the block interval, finishing allowance;

⑶: 3rd blocks for some continuous finishing path, counting the roughing path without being executed actually when executing G71.

According to the finishing path, the finishing allowance, the path of tool infeed and tool retract, the system automatically counts the path of roughing，the tool cuts the workpiece in paralleling with Z, and the roughing is completed by multiple executing the cutting cycle tool infeed→ cutting→tool retraction. The starting point and the end point

are the same one. The command is applied to the formed roughing of non-formed rod.

Command format： G71 U（Δd） R（e） F__ S__ T__； (1)

G71 P（ns） Q（nf） U（Δu） W（Δw）； (2)

N（ns） . . . . . ；
. . . . . ；
. . . . F；
. . . . S；
. . . . (3)
N（nf）. . . . . ；

Command specifications：

(1) ns～nf blocks in programming must be followed G71 blocks. If they are in the front of G71 blocks, the system automatically searches and executes ns～nf blocks, and then executes the next program following nf block after they are executed, which causes the system executes ns～nf blocks repetitively;

(2) ns～nf blocks are used to count the roughing path and the blocks are not executed when G71 is executed. F, S, T commands of ns～nf blocks are invalid when G71 is executed, at the moment, F, S, T commands of G71 blocks are valid. F, S, T of ns～nf blocks are valid when executing ns～nf to command G70 finishing cycle;

(3) For G71 (I type), ns block is only G00, G01 which has no Z (W) in Group 01, otherwise, the system considers it G71 (Ⅱ type) machining;

(4) X and Z dimensions must be changed monotonously (always increasing or reducing) for the finishing path;

(5) In ns～nf blocks, there are only G commands: G01, G02, G03, G04, G96, G97, G98, G99, G40, G41,G42 and the system cannot call subprograms(M98/M99);

(6) G96, G97, G98, G99, G40, G41, G42 are invalid in G71 and valid in G70, G96, G97, G98,

(7) When G71 is executed, the system can stop the automatic run and manual traverse, but return to the position before manual traversing when G71 is executed again, otherwise, the following path will be wrong;

(8) When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path;

(9) △d，△u are specified by the same U and different with or without being specified P,Q commands;

(10) G71 cannot be executed in MDI, otherwise, the system alarms;

Relevant definitions:

Finishing path	As Fig. 2-34, Part 3 of G71(ns～nf block)defines the finishing path, and the starting point of finishing path (starting point of ns block)is the same these of starting point and end point of G71, called A point; the first block of finishing path(ns block)is used to X rapid traversing or tool infeed, and the end point of finishing path is called to B point; the end point of finishing path(end point of nf block)is called to C point. The finishing path is A→B→C

Roughing path	The finishing path is the one after offsetting the finishing allowance（Δu, Δw）and is the path contour formed by executing G71. A, B, C point of finishing path after offset corresponds separately to A', B', C' point of roughing path, and the final continuous cutting path of G71 is B'→C' point
Δd	It is each travel（radius value）of X tool infeed in roughing without sign symbols, and the direction of tool infeed is defined by move direction of ns block. The command value Δd is reserved after executing U（Δd）and the value of NO.5132 is rewritten. The value of system parameter NO.5132 is regarded as the travel of tool infeed when U（Δd）is not input
e	It is travel（radius value）of X tool retraction in roughing（radius value）without sign symbols, and the direction of tool retraction is opposite to that of tool infeed, the command value e is reserved and the value of system parameter NO.5133 is rewritten after R（e）is executed. The value of system parameter NO.5133 is regarded as the travel of tool retraction when R（e）is not input
ns	Block number of the first block of finishing path
nf	Block number of the last block of finishing path
Δu	X finishing allowance range is as the following table (diameter) with sign symbols. X coordinate offset of roughing path compared to finishing path, i.e. the different value of X absolute coordinates between A' and A. The system defaults Δu=0 when U（Δu）is not input, i.e. there is no X finishing allowance for roughing cycle
Δw	Z finishing allowance range is as the following table (diameter) with sign symbols. X coordinate offset of roughing path compared to finishing path, i.e. the different value of X absolute coordinates between A' and A. The system defaults Δw=0 when U（Δw）is not input, i.e. there is no Z finishing allowance for roughing cycle
F	Cutting feedrate; S: Spindle speed; T: Tool number, tool offset number
M, S, T, F	They can be specified in the first G71 or the second ones or program ns～nf. M, S, T, F functions of M, S, T, F blocks are invalid in G71, and they are valid in G70 finishing blocks

Address	Incremental system	metric（mm）input	inch(inch) input
U（Δd）	ISB system	0.001~99999.999	0.0001~9999.9999
	ISC system	0.0001~9999.9999	0.00001~999.99999
R（e）	ISB system	0～99999.999	0～9999.9999
	ISC system	0～9999.9999	0～999.99999
U（Δu）	ISB system	-99999.999～99999.999	-9999.9999～9999.9999
	ISC system	-9999.9999～9999.9999	-999.99999～999.99999
W（Δw）	ISB system	-99999.999～99999.999	-9999.9999～9999.9999
	ISC system	-9999.9999～9999.9999	-999.99999～999.99999
P（ns）	ISC system	1～99999	1～99999
	ISC system	1～99999	1～99999
Q（nf）	ISC system	1～99999	1～99999
	ISC system	1～99999	1～99999

Execution process: as Fig. 2-36.

① X rapidly traverses to A' from A point, X travel is Δu, and Z travel is Δw

② X moves from A'is Δd(tool infeed), ns block is for tool infeed at rapid traverse speed with G0, is for tool infeed at feedrate F with G71, and its direction of tool infeed is that of A→B point;

③ Z executes the cutting feeds to the roughing path, and its direction is the same that of Z coordinate A→B point;

④ X, Z execute the tool retraction e (45°straight line)at feedrate, the directions of tool retraction is opposite to that of too infeed;

⑤ Z rapidly retracts at rapid traverse speed to the position which is the same that of Z coordinate;

⑥ After executing X tool infeed (Δd+e)again, the end point of traversing tool is still on the middle point of straight line between A' and B'(the tool does not reach or exceed B'), and after executing the tool infeed (Δd+e)again, execute ③; after executing the tool infeed (Δd+e)again, the end point of tool traversing reaches B' point or exceeds the straight line between A'→B' point and X executes the tool infeed to B' point, and then the next step is executed;

⑦ Cutting feed from B' to C' point along the roughing path;

⑧ Rapid traverse to A from C' point and the program jumps to the next clock following nf block after G71 cycle is ended.

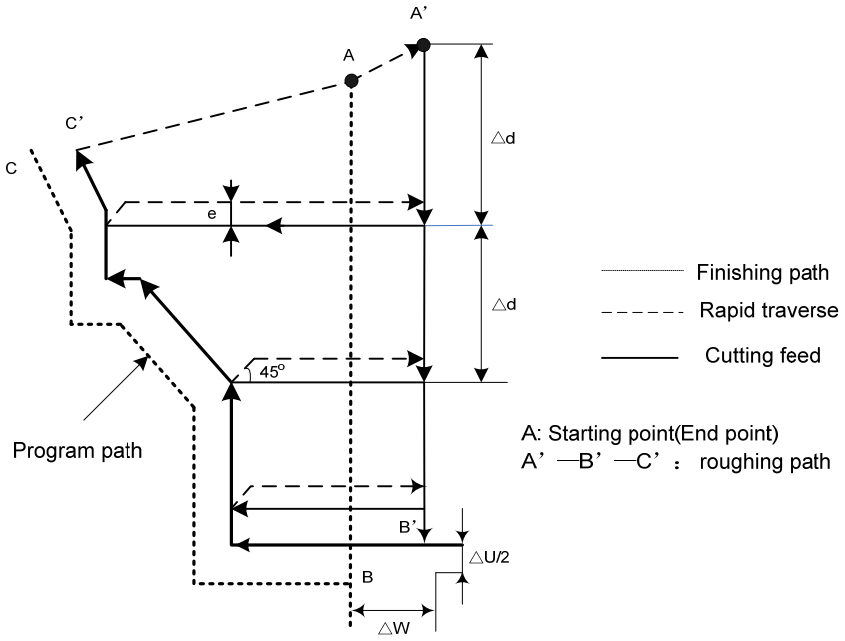


Fig. 2-36 G71 cycle path

Coordinate offset direction with finishing allowance:

Δu, Δw define the coordinates offset and its direction of finishing, and their sign symbols are as follows Fig. 2-35: B→C for finishing path, B'→C' for roughing path and A is the tool starting point

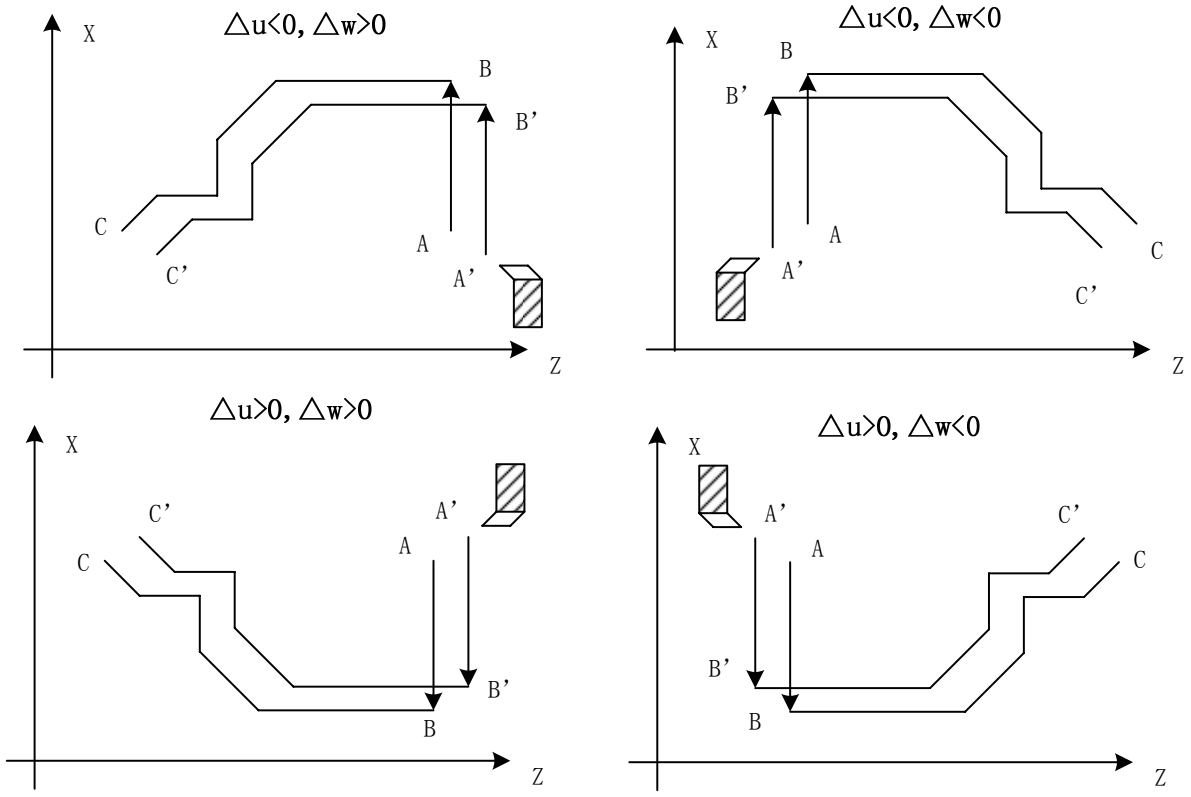


Fig.2-37

Example: Fig. 2-38

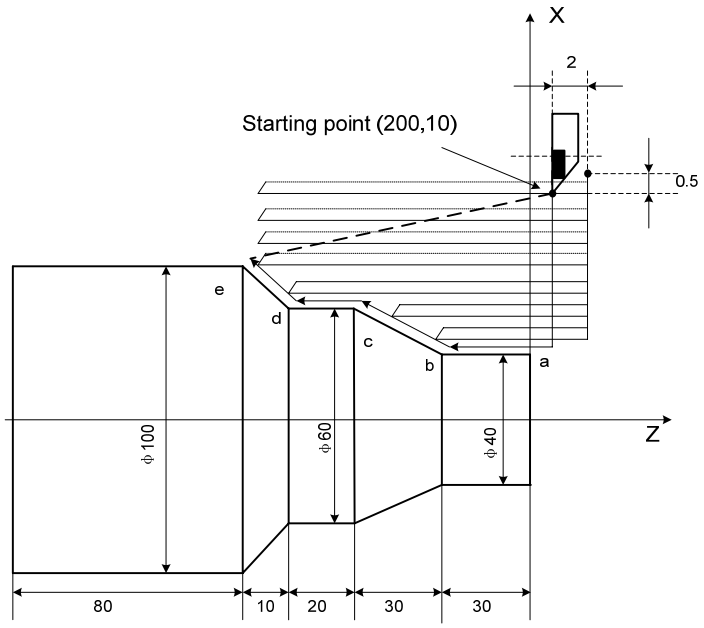


Fig.2-38

Program：O0004；

G00 X200 Z10 M3 S800； （Spindle clockwise with 800 rev/min）

G71 U2 R1 F200； （Cutting depth each time 4mm，tool retraction [in diameter]）

```
G71 P80 Q120 U0.5 W0.2；   （roughing a---e，X machining allowance 0.5mm , Z  0.2mm）
N80 G00 X40 S1200；        （Positioning）
G01 Z-30 F100 ；           （a→b）
X60 W-30;                  （b→c）
W-20;                      （c→d）
N120 X100 W-10;            （d→e）
G70 P80 Q120;              （a---e blocks for finishing path）
M30;                       （End of block）
```

a→b→c→d→e blocks for finishing path

G71 supports continuous grooving machining:

Direction of the shape in the 2nd axis of the level (X axis in ZX level) is not necessary to monotonous rise or fall, and there may be up to 10 groovings, which is shown below:

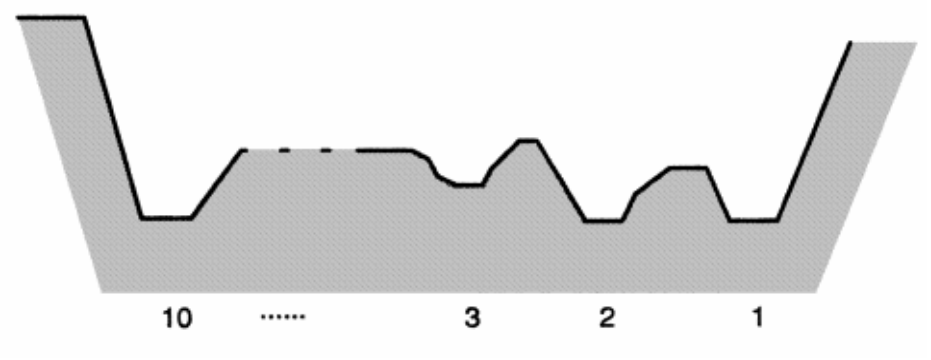
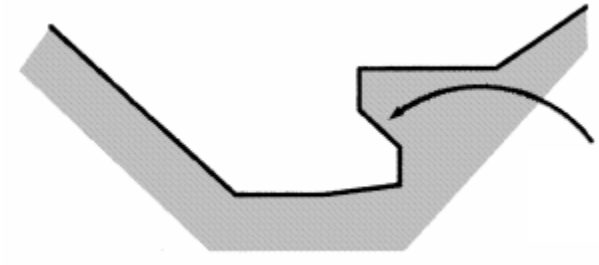


Fig. 2-39

But, external contour along Z must mononously rise or fall, and the following contour cannot be machined:



Monontone change is not observed along the Z axis

Fig. 2-40

The first tool must be vertical: the machining can be executed when the shape along Z changes mononously, which is shown below:

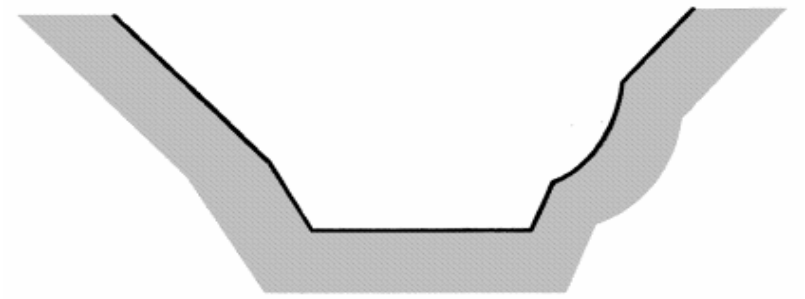


Fig. 2-41

The tool retraction should be executed after turning, and the retraction amount is specified by R (e) or No 5133, which is shown below:

e（set by a parameter）

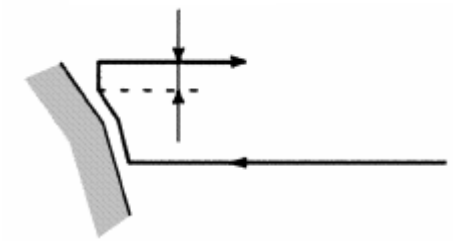


Fig. 2-42

Execution process sketch:

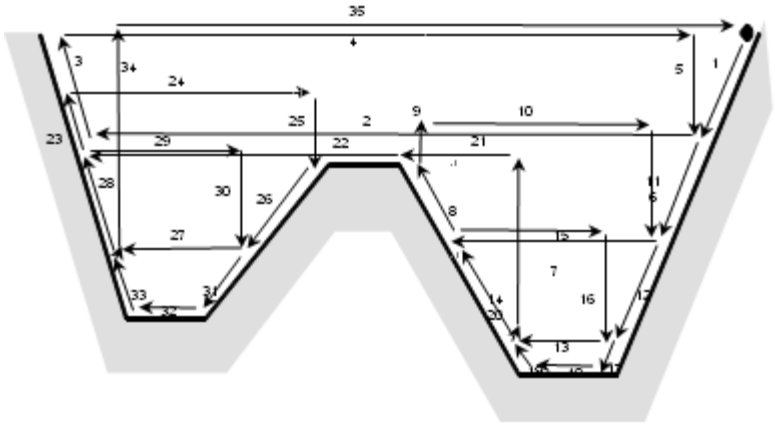


Fig. 2-43

Note 1: For grooving, X (U), Z(W) must be specified, and W0 is done when Z does not move.
Note 2: For grooving, the finishinig allowance is specified to X direction, is invalid for Z direction.

Note 3: For grooving, the tool retraction amount is left to make the tool approach the workpiece(Label 25, 26) with G1 speed after the current grooving is done to execute the next grooving. When the retraction amount is 0 or the left distance is less than retraction amount, the tool approaches the workpiece with G1 speed.

Note 4: For grooving, the finishing path (ns～nf block), Z dimension must mononously change (always increase or decrease)

Note 5: For G71 II type, when there is arc in finishing path (ns～nf), # 3410 parameter (the arc radius permits error) cannot be non-zero, i.e., the permitting function of arc radius error cannot be activated.

Note 6: Radius error is irrelevant to cutting allowance, and radius error is permitted and checks whether the alarm occurs.

2.15.2 Radial Roughing Cycle G72

Command function：G72 is divided into three parts:

(1) 1st blocks for defining the travels of tool infeed and tool retraction, the cutting speed, the spindle speed and the tool function in roughing;

⑵ 2nd blocks for defining the block interval, finishing allowance;

⑶ 3rd blocks for some continuous finishing path, counting the roughing path without being executed actually when G72 is executed.

According to the finishing path, the finishing allowance, the path of tool infeed and retract tool, the system automatically counts the path of roughing，the tool cuts the workpiece in paralleling with Z, and the roughing is completed by multiple executing the cutting cycle tool infeed→cutting feed→tool retraction. The starting point and the end point of G72 are the same one. The command is applied to the formed roughing of non-formed rod.

Command format：G72 W（Δd） R（e） F__ S__ T__； (1)

G72 P（ns） Q（nf） U（Δu） W（Δw）； (2)

N__（ns） . . . . . ;
. . . . . ;
. . . . F;
. . . . S;
. . . . ; (3)
.

N__（nf）.....;

Command specifications:

1. ns～nf blocks in programming must be followed G72 blocks. If they are in the front of G72 blocks, the system automatically searches and executes ns～nf blocks, and then executes the next program following nf block after they are executed, which causes the system executes ns～nf blocks repetitively;
2. ns～nf blocks are used for counting the roughing path and the blocks are not executed when G72 is executed. F, S, T commands of ns～nf blocks are invalid when G72 is executed, at the moment, F, S, T commands of G72 blocks are valid. F, S, T of ns～nf blocks are valid when executing ns～nf to command G70 finishing cycle;
3. There are G00,G01 without the word X(U) in ns block, otherwise the system alarms;
4. X,Z dimensions in finishing path(ns～nf blocks) must be changed monotonously (always increasing or reducing) for the finishing path;
5. In ns～nf blocks, there are only G commands: G01, G02, G03, G04, G96, G97, G98, G99, G40, G41,G42 and the system cannot call subprograms(M98/M99);
6. G96, G97, G98, G99, G40, G41, G42 are invalid in G72 and valid in G70;
7. When G72 is executed, the system can stop the automatic run and manual traverse, but return to the position before manual traversing when G72 is executed again, otherwise, the following path will be wrong;
8. When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path;
9. △d，△u are specified by the same U and different with or without being specified P,Q commands;
10. G72 cannot be executed in MDI, otherwise, the system alarms.

Relevant definitions:

Finishing path	the above-mentioned Part⑶ of G71(ns～nf block)defines the finishing path, and

	the starting point of finishing path (i.e. starting point of ns block)is the same these of starting point and end point of G72, called A point; the first block of finishing path(ns block)is used for Z rapid traversing or cutting feed, and the end point of finishing path is called to B point; the end point of finishing path(end point of nf block)is called to C point. The finishing path is A→B→C.
Roughing path	The finishing path is the one after offsetting the finishing allowance（Δu, Δw）and is the path contour formed by executing G72. A, B, C point of finishing path after offset corresponds separately to A', B', C'point of roughing path, and the final continuous cutting path of G72 is B'→C' point.
Δd	It is each travel of Z tool infeed in roughing without sign symbols, and the direction of tool infeed is defined by move direction of ns block. Δd is reserved after the system executes W（Δd）and NO.5132 value is modified. The value of system parameter NO.051 is regarded as the travel of tool infeed when W（Δd）is not input.
e	It is each travel of Z tool infeed in roughing without sign symbols, and the direction of tool retraction is opposite to that of tool infeed; after R(e) is executed, e value e is reserved and the system modifies No.5133 value. The value of system parameter NO.5133 is regarded as the travel of tool retraction when R（e）is not input.
ns	Block number of the first block of finishing path.
nf	Block number of the last block of finishing path.
Δu	X finishing allowance in roughing, (X coordinate offset of roughing path compared to finishing path, i.e. the different value of X absolute coordinate between A'and A, diameter value with sign symbols).
Δw	Z finishing allowance in roughing, its value: -9999.999~9999.999 (Z coordinate offset of roughing path compared to finishing path, i.e. the different value of X absolute coordinates between A' and A, with sign symbols).
F	Cutting feedrate; S: Spindle speed; T: Tool number, tool offset number.
M, S, T, F	They can be specified in the first G72 or the second ones or program ns～nf. M, S, T, F functions of M, S, T, F blocks are invalid in G72, and they are valid in G70 finishing blocks.

Address	Incremental system	Metric（mm）input	Inch (inch) input
W（Δd）	ISB system	0.001~99999.999	0.0001~9999.9999
	ISC system	0.0001~9999.9999	0.00001~999.99999
R（e）	ISB system	0～99999.999	0～9999.9999
	ISC system	0～9999.9999	0～999.99999
U（Δu）	ISB system	-99999.999～99999.999	-9999.9999～9999.9999
	ISC system	-9999.9999～9999.9999	-999.99999～999.99999
W（Δw）	ISB system	-99999.999～99999.999	-9999.9999～9999.9999
	ISC system	-9999.9999～9999.9999	-999.99999～999.99999
P（ns）	ISC system	1～99999	1～99999
	ISC system	1～99999	1～99999
Q（nf）	ISC system	1～99999	1～99999

	ISC system	1～99999	1～99999

Execution process： Fig. 2-44

① X rapidly traverses to A' from A point, X travel is Δu, and Z travel is Δw;

② X moves from m A'is Δd(tool infeed), ns block is for tool infeed at rapid traverse speed with G0, is for tool infeed at G72feedrate F in G1, and its direction of tool infeed is that of A→B point;

③ X executes the cutting feeds to the roughing path, and its direction is the same that of X coordinate B→C point;

④ X, Z execute the tool retraction e (45°straight line)at feedrate, the directions of tool retraction is opposite to that of tool infeed ;

⑤ X rapidly retracts at rapid traverse speed to the position which is the same that of Z coordinate;

⑥ After Z tool infeed (Δd+e)again is executed, the end point of traversing tool is still on the middle point of straight line between A' and B'(the tool does not reach or exceed B'), and after Z executes the tool infeed (Δd+e)again, ③ is executed；after the tool infeed (Δd+e) is executed again, the end point of tool traversing reaches B' point or exceeds the straight line between A'→B' point and Z executes the tool infeed to B' point, and then the next step is executed;

⑦ Cutting feed from B' to C' point along the roughing path;

⑧ Rapidly traverse to A from C' point and the program jumps to the next clock following nf block after G71 cycle is completed.

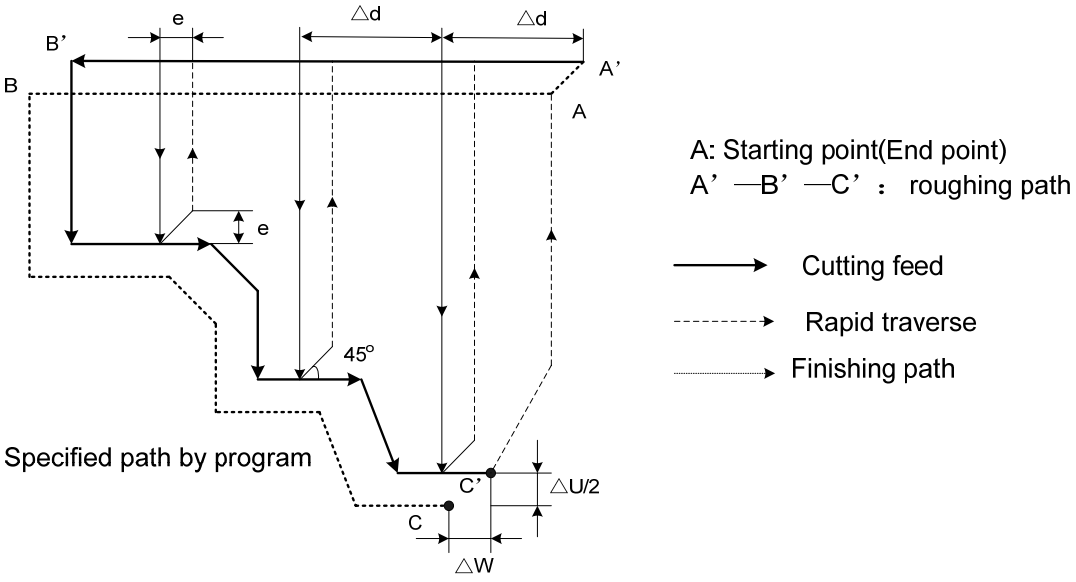


Fig. 2-44

Coordinate offset direction with finishing allowance:

Δu, Δw define the coordinates offset and its direction of finishing, and their sign symbols are as follows Fig. 2-45: B→C for finishing path, B'→C' for roughing path and A is the starting point.

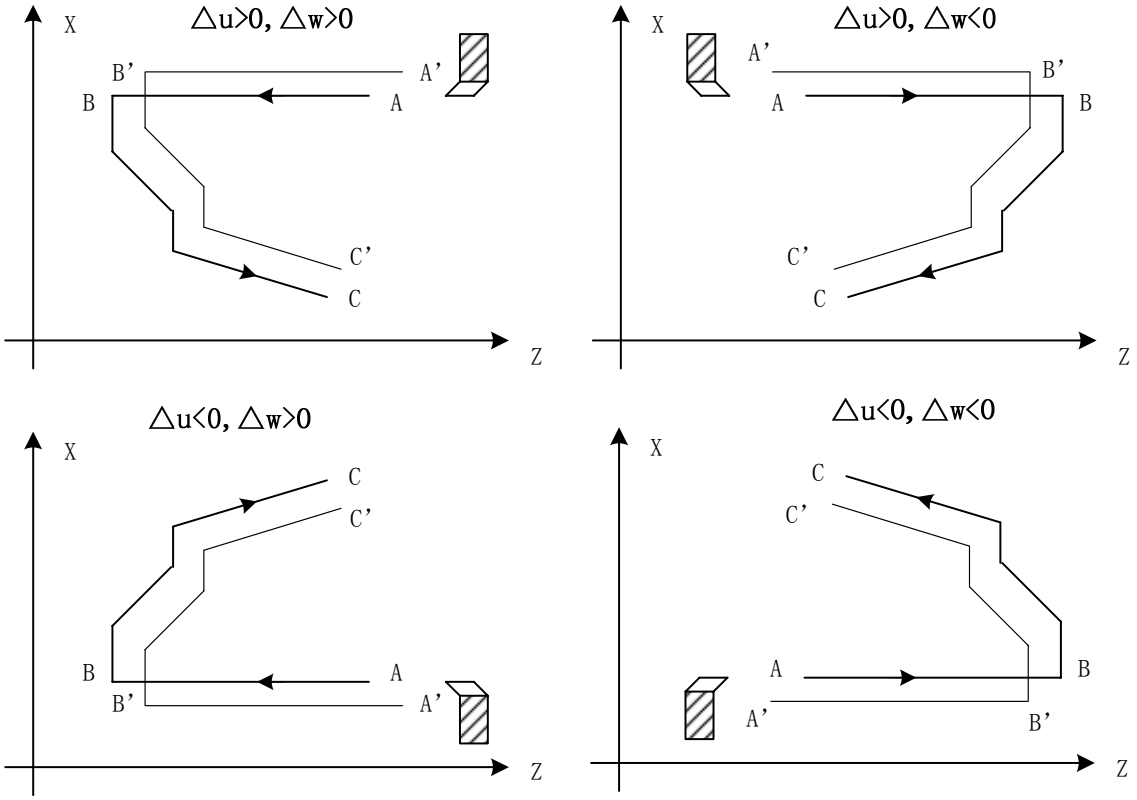


Fig.2-45

Example: Fig. 2-46

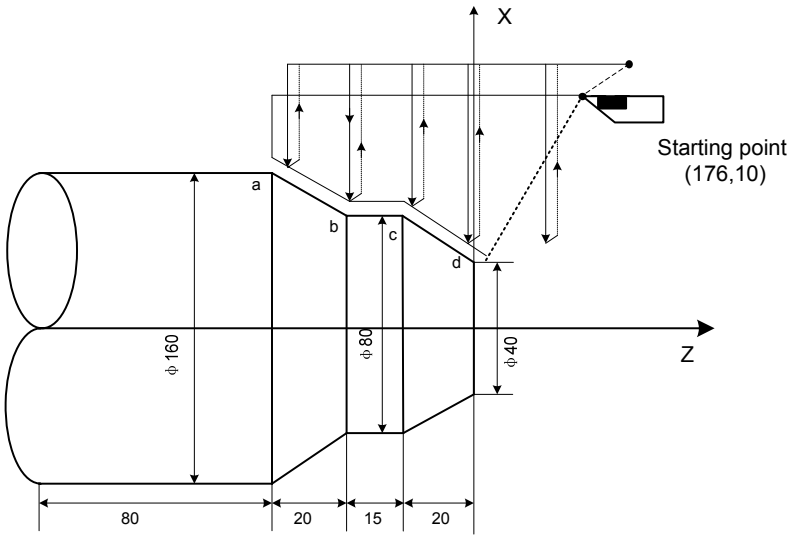


Fig.2-46

Program：

O0005；

G00 X176 Z10 M03 S500 （Change No.2 tool and execute its compensation, spindle rotation with 500 rev/min）

G72 W2.0 R0.5 F300； （Tool infeed 2mm, tool retraction 2mm）

G72 P10 Q20 U0.2 W0.1； （Roughing a--d，X roughing allowance 0.2mm and Z 0.1mm）

N10 G00 Z-55 S800 ； （Rapid traverse）

```
G01 X160 F120;        (Infeed to a point)
X80 W20;              (Machining a—b)
W15;                  (Machining b—c)      } Blocks for finishing path
N20 X40 W20 ;         (Machining c—d)
G70 P050 Q090 M30;    (Finishing a—d)
```


2.15.3 Closed Cutting Cycle G73

Command functions: G73 is divided into three parts:

(1) Blocks for defining the travels of tool infeed and tool retraction, the cutting speed, the spindle speed and the tool function when roughing;

(2) Blocks for defining the block interval, finishing allowance;

(3) Blocks for some continuous finishing path, counting the roughing path without being executed actually when executing G73.

According to the finishing allowance, the travel of tool retraction and the cutting times, the system automatically counts the travel of roughing offset，the travel of each tool infeed and the path of roughing, the path of each cutting is the offset travel of finishing path, the cutting path approaches gradually the finishing one, and last cutting path is the finishing one according to the finishing allowance. The starting point and end point of G73 are the same one, and G73 is applied to roughing for the formed rod. G73 is non-modal and its path is as Fig.2-40.

Command forma：

```
G73 U（Δi） W （Δk） R （d） F__ S__ T__;          (1)
G73 P（ns） Q（nf） U（Δu） W（Δw）;               (2)
N__（ns） . . . . . ;
. . . . . ;
. . . . F;
. . . . S;
. . . . ;                                          (3)
.
N__（nf）. . . . . ;
```

Command specifications：

1. ns～nf blocks in programming must be followed G73 blocks. If they are in the front of G73 blocks, the system automatically searches and executes ns～nf blocks, and then executes the next program following nf block after they are executed, which causes the system executes ns～nf blocks repetitively.
2. ns～nf blocks are used for counting the roughing path and the blocks are not executed when G73 is executed. F, S, T commands of ns～nf blocks are invalid when G71 is executed, at the moment, F, S, T commands of G73 blocks are valid. F, S, T of ns～nf

blocks are valid when executing ns～nf to command G70 finishing cycle.

3. There are only G00, G01 in ns block.
4. In ns～nf blocks, there are only G commands:G00, G01, G02, G03, G04, G96, G97, G98, G99, G40, G41,G42 and the system cannot call subprograms(M98/M99)
5. G96, G97, G98, G99, G40, G41, G42 are invalid in G73 and valid in G70.
6. When G73 is executed, the system can stop the automatic run and manual traverse, but return to the position before manual traversing when G73 is executed again, otherwise, the following path will be wrong.
7. When the system is executing the feed hold or single block, the program pauses after the system has executed end point of current path.
8. △i，△u are specified by the same U and Δk，Δw are specified by the same U, and they are different with or without being specified P, Q commands.
9. G73 cannot be executed in MDI, otherwise, the system alarms.
10. Z must be the monotonous in the cycle body specified by P and Q. Z tool retraction and finishing allowance are set to 0 when the system executes X non-monotonous workpiece. When No. 5102 Bit0 (MRI) is set to 1, the system does not alarm.
11. When the programming is executed, and the initial positioning point retreats one tool infeed value in the direction of cutting but the result is in the contour range, the dry run is executed to observe whether its own path of the system has overcutting because the tool retraction direction is the same that of tool infeed in programming state.

Relevant definitions:

Finishing path	The above-mentioned Part 3 of G73**(**ns～nf block)defines the finishing path, and the starting point of finishing path (start point of ns block)is the same these of starting point and end point of G73, called A point; the end point of the first block of finishing path(ns block)is called B point; the end point of finishing path(end point of nf block)is called C point. The finishing path is A→B→C.
Roughing path	It is one group of offset path of finishing one, and the roughing path times are the same that of cutting. After the coordinates offset, A, B, C of finishing path separately corresponds to A_n, B_n, C_n of roughing path(n is the cutting times, the first cutting path is A_1, B_1, C_1 and the last one is A_d, B_d, C_d). The coordinates offset value of the first cutting compared to finishing path is （Δi×2+Δu，Δw+Δk）(diameter programming）, the coordinates offset value of the last cutting compared to finishing path is（Δu，Δw），the coordinates offset value of each cutting compared to the previous one is（Δi×2/d-1，Δk/d-1）.
Δi	Travel of X tool retraction in roughing is the following table（radius value with sign symbols）, Δi is equal to X coordinate offset value（radius value）of A_1 point compared to A_d point. The X total cutting travel(radius value) is equal to |Δi| in roughing, and X cutting direction is opposite to the sign symbol of Δi: Δi＞0, cut in X negative direction in roughing. It is reserved after Δi command value is executed and the system rewrites No.5135 value. NO.5135 value is regarded as the travel of X tool retraction of roughing when U（Δi）is not input.
Δk	Travel of Z tool retraction in roughing is the following table（radius value with sign symbols）, Δk is equal to X coordinate offset value（radius value）of A1point compared to A_d point. The Z total cutting travel(radius value) is equal to |Δk| in

	roughing, and Z cutting direction is opposite to the sign symbol of Δk: Δk＞0, cut in Z negative direction in roughing. It is reserved after Δk command value is executed and the system rewrites No.5136 value. NO.5136 value is regarded as the travel of X tool retraction of roughing when W（Δk）is not input.
d	It is the cutting times and its range is referred to the following table. R5 means the closed cutting cycle is completed by 5 times cutting. R（d）is reserved after it is executed and the system rewrites NO.5137. The value of system parameter NO.5137 is regarded as the cutting times when R（d is not input.
ns	Block number of the first block of finishing path.
nf	Block number of the last block of finishing path.
Δu	It is X finishing allowance as the following table (diameter value with sign symbols) and is the X coordinate offset of roughing contour compared to finishing path, i.e. the different value of X absolute coordinates of A_1 compared to A. Δu＞0，it is the offset of the last X positive roughing path compared to finishing path. The system defaults Δu=0 when U（Δu） is not input, i.e. there is no X finishing allowance for roughing cycle.
Δw	It is Z finishing allowance as the following table -99.999~99.999 (unit: mm) and is the Z coordinate offset of roughing contour compared to finishing path, i.e. the different value of Z absolute coordinate of A_1 compared to A. Δw＞0，it is the offset of the last roughing path compared to finishing path in Z positive direction. The system defaults Δw=0 when W（Δw）is not input, i.e. there is no Z finishing allowance for roughing cycle.
F	Feedrate; S: Spindle speed; T: Tool number, tool offset number.
M, S, T, F	They can be specified in the first G73 or the second ones or program ns～nf. M, S, T, F functions of M, S, T, F blocks are invalid in G73, and they are valid in G70 finishing blocks.

Address	Incremental system	Metric（mm）input	Inch (inch) input
U（Δi）	ISB system	-99999.999～99999.999 mm	-9999.9999～9999.9999 inch
	ISC system	-9999.9999～9999.9999 mm	-999.99999～999.99999 inch
W（Δk）	ISB system	-99999.999～99999.999 mm	-9999.9999～9999.9999 inch
	ISC system	-9999.9999～9999.9999 mm	-999.99999～999.99999 inch
R（d）	ISB, ISC	1～999（times）	1～999（times）
U (Δu）	ISB system	-99999.999～99999.999 mm	-9999.9999～9999.9999 inch
	ISC system	-9999.9999～9999.9999 mm	-999.99999～999.99999 inch
W（Δw）	ISB system	-99999.999～99999.999 mm	-9999.9999～9999.9999 inch
	ISC system	-9999.9999～9999.9999 mm	-999.99999～999.99999 inch

Execution process:(Fig. 2-40).

① A→A_1：Rapid traverse；

② First roughing A_1→B_1→C_1 ：

A_1→B_1：Rapid traverse speed in ns block in G0, cutting feedrate specified by G73 in ns block in G1；

$B_1 \rightarrow C_1$：Cutting feed.

③ $C_1 \rightarrow A_2$：Rapid traverse；

④ Second roughing $A_2 \rightarrow B_2 \rightarrow C_2$ ：

$A_2 \rightarrow B_2$：Rapid traverse speed in ns block in G0, cutting feedrate specified by G73 in ns block in G1;

$B_2 \rightarrow C_2$：Cutting feed.

⑤ $C_2 \rightarrow A_3$：rapid traverse；

.....

No. n times roughing，$A_n \rightarrow B_n \rightarrow C_n$ ：

$A_n \rightarrow B_n$：ns Rapid traverse speed in ns block in G0, cutting feedrate specified by G73 in ns block in G1;

$B_n \rightarrow C_n$：Cutting feed.

$C_n \rightarrow A_{n+1}$：Rapid traverse;

.....

Last roughing，$A_d \rightarrow B_d \rightarrow C_d$ ：

$A_d \rightarrow B_d$：Rapid traverse speed in ns block in G0, cutting feedrate specified by G73 in ns block in G1;

$B_d \rightarrow C_d$：Cutting feed.

$C_d \rightarrow A$：Rapid traverse to starting point;

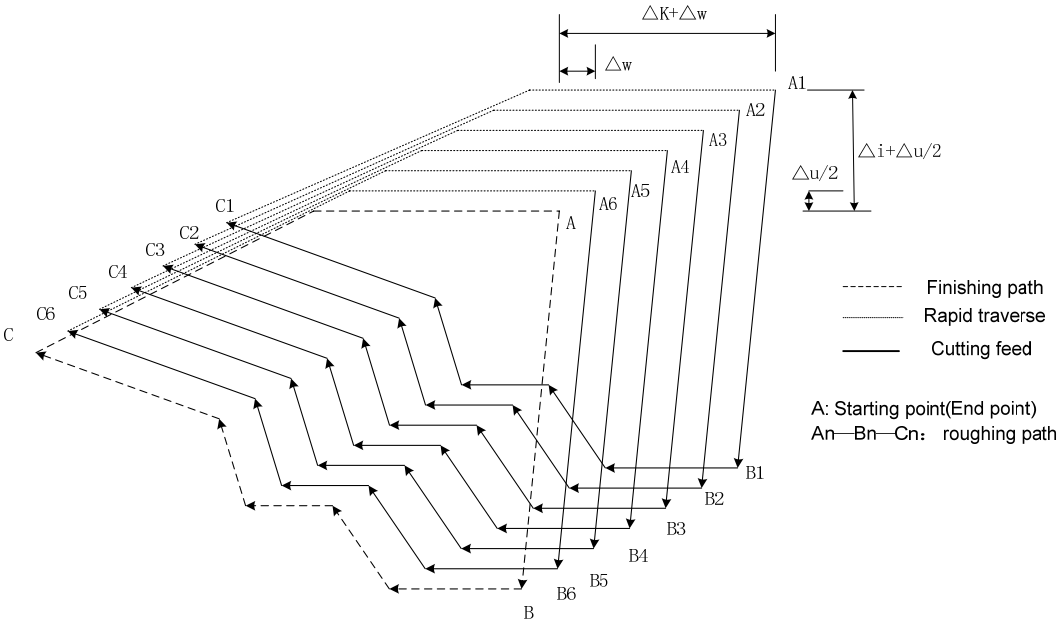


Fig. 2-47 G73 run path

Coordinate offset direction with finishing allowance:

Δi, Δk define the coordinates offset and its direction of roughing, Δu, Δw define the coordinates offset and cut-in direction in finishing; Δi, Δk, Δu, Δw can consist of many groups. Generally, the sign symbols of Δi and Δu are consistent, the sign symbols of Δk and Δw are consistent, there are four kinds of combination as Fig. 3-48,A for start-up tool point, B→C for workpiece contour, B'→C'for roughing contour and B''→C'' for finishing path.

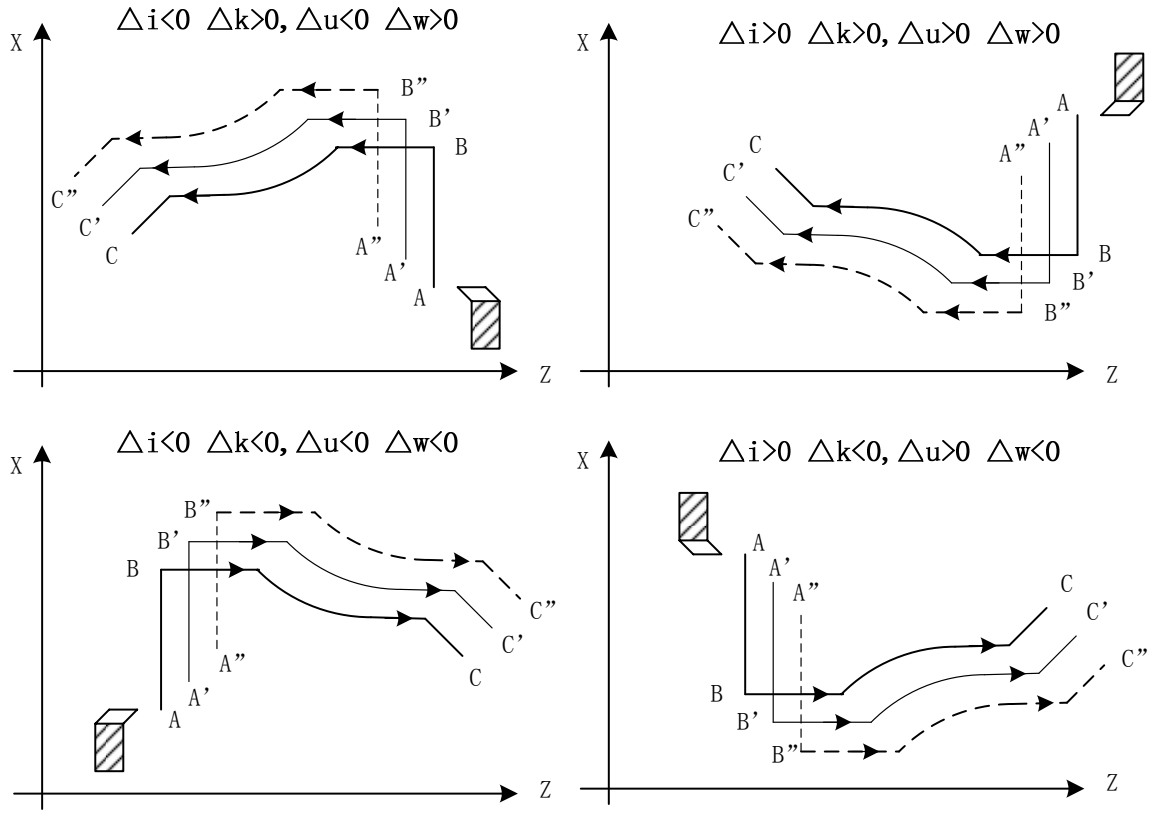


Fig.2-48

Example: Fig. 2-49

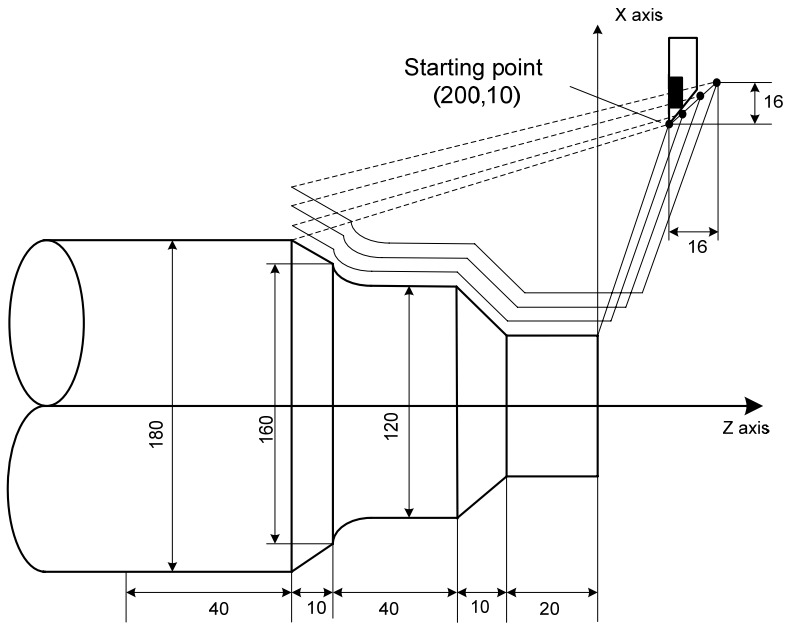


Fig.2-49

Program：O0006；

G99 G00 X200 Z10 M03 S500; （Specify feedrate per rev and position starting point and start spindle）

G73 U1.0 W1.0 R3 ; （X tool retraction with 2mm, Z 1mm）

```
G73 P14 Q19 U0.5 W0.3 F0.3 ;      （X roughing with 0.5 allowance and Z 0.mm）

N14 G00 X80 W-40 ;
G01 W-20 F0.15 S600 ;
X120 W-10 ;                        } Blocks for finishing
W-20 ;
G02 X160 W-20 R20 ;
N19 G01 X180 W-10 ;
G70 P14 Q19 M30;                   } （Finishing）
```

2.15.4 Finishing Cycle G70

Command function：The tool executes the finishing of workpiece from starting point along with the finishing path defined by ns～nf blocks. After executing G71, G72 or G73 to roughing, execute G70 to finishing and single cutting of finishing allowance is completed. The tool returns to starting point and execute the next block following G70 block after G70 cycle is completed.

Command format：G70 P（ns） Q（nf）；

Command specifications:

1. ns：Block number of the first block of finishing path
   nf: Block number of the last block of finishing path.
   G70 path is defined by programmed one of ns～nf blocks. Relationships of relative position of ns, nf block in G70～G73 blocks are as follows:

```
. . . . . .
G71/G72/G73 ……;
N__（ns） . . . . . .
 . . . . . .
      · F
      · S                  } Blocks for finishing path
      .
      .
N__（nf）……
     . . .
G70 P（ns） Q（nf）;
      . . .
      . . .
```

2. G70 is compiled following ns～nf blocks. If they are in the front of G71 blocks, the system automatically searches and executes ns～nf blocks, and then executes the next program following nf block after they are executed, which causes the system executes ns～nf blocks repetitively.
3. F, S, T in ns～nf blocks are valid when executing ns～nf to command G70 finishing cycle.
4. G96, G97, G98, G99, G40, G41, G42 are valid in G70;
5. When G70 is executed, the system can stop the automatic run and manual traverse, but return to the position before manual traversing when G70 is executed again, otherwise, the following path will be wrong.
6. When the system is executing the single block, the program pauses after the system has

executed end point of current path.

7. G70 cannot be executed in MDI mode, otherwise, the system alarms.

2.15.5 Axial Grooving Multiple Cycle G74

Command function: Axial (X) tool infeed cycle compounds radial discontinuous cutting cycle: Tool infeeds from starting point in radial direction(Z), retracts, infeeds again, and again and again, and last tool retracts in axial direction, and retracts to the Z position in radial direction, which is called one radial cutting cycle; tool infeeds in axial direction and execute the next radial cutting cycle; cut to end point of cutting, and then return to starting point (starting point and end point are the same one in G74), which is called one radial grooving compound cycle. Directions of axial tool infeed and radial tool infeed are defined by relative position between end point X（U）Z（W） and starting point of cutting. The command is used to machine radial loop groove or column surface by radial discontinuously cutting, breaking stock and stock removal.

Command format： G74 R（e）;

G74 X（U）__ Z（W）__ P（Δi） Q（Δk） R（Δd） F__;

Command specifications：

（1）The cycle movement is executed by Z（W）and P（Δk）blocks of G74, and the movement is not executed if only “G74 R（e）；” block is executed;

（2）Δd and e are specified by the same address and whether there are Z（W）and P（Δk） word or not in blocks to distinguish them;

（3）The tool can stop in Auto mode and traverse in Manual mode when G74 is executed, but the tool must return to the position before executing in Manual mode when G74 is executed again, otherwise the following path will be wrong.

（4）When the single block is running, programs pauses after each axial cutting cycle is completed.

（5）R（Δd）must be omitted in blind hole cutting, and so there is no distance of tool retraction when the tool cuts to axial end point

Relevant definitions：

Starting point of axial cutting cycle	Starting position of axial tool infeed for each axial cutting cycle, defining with A_n(n=1,2,3……), Z coordinate of A_n is the same that of starting point A, the different value of X coordinate between A_n and A_{n-1} is Δi. The starting point A_1 of the first axial cutting cycle is the same as the starting point A, and the X coordinate of starting point (A_f) of the last axial cutting cycle is the same that of cutting end point.
End point of axial tool infeed	Starting position of axial tool infeed for each axial cutting cycle, defining with B_n(n=1,2,3……), Z coordinate of B_n is the same that of cutting end point, X coordinate of B_n is the same that of A_n , and the end point (B_f) of the last axial tool infeed is the same that of cutting end point.
End point of radius tool retraction	End position of radius tool infeed(travel of tool infeed is Δd) after each axial cutting cycle reaches the end point of axial tool infeed, defining with C_n(n=1,2,3……), Z coordinate of C_n is the same that of cutting end point, and the different value of X

	coordinate between C_n and A_n is Δd;
End point of axial cutting cycle	End position of axial tool retraction from the end point of radius tool retraction, defining with D_n(n=1,2,3……), Z coordinate of D_n is the same that of starting point, X coordinate of D_n is the same that of C_n (the different value of X coordinate between it and A_n is Δd);
Cutting end point	It is defined by X（U）__ Z（W）___ ,and is the end point B_f of last axial tool infeed.
R（e）	It is the travel of tool retraction after each axial(Z) tool infeed without sign symbols as the following table. The command value is reserved after executing R（e）and the value of NO.5139 is rewritten. The value of NO.5139 is regarded as the travel of tool retraction when R（e）is not input.
X	X absolute coordinate value of cutting end point B_f (unit: mm)
U	Different value of X absolute coordinate between cutting end point B_f and starting point.
Z	Z absolute coordinate value of cutting end point B_f (unit: mm).
W	Different value of Z absolute coordinate between cutting end point B_f and starting point.
P（Δi）	Travel of radial(X) cutting for each axial cutting cycle without sign symbols, and the value range is referred to the following table.
Q（Δi）	Travel of Z discontinuous tool infeed without sign symbols in axial(Z) cutting, and the value range is referred to the following table.
R（Δd）	Travel (radius value)of radial (X) tool retraction after cutting to end point of axial cutting. The value range is referred to the following table. The radial (X) tool retraction is 0 when R（Δd）is omitted and the system defaults the axial cutting end point. The radial (X) tool retraction is 0 when P（Δi） is omitted.

Address	Incremental system	metric（mm） input	Inch (inch) input
P（Δi）	ISB system	0~99999999（unit:0.001mm）	0~99999999（unit:0.0001inch）
Q（Δk）	ISC system	0~99999999（unit:0.0001mm）	0~99999999（unit:0.00001inch）
R（e）	ISB system	0～99999.999mm	0～9999.9999 inch
R（Δd）	ISC system	0～9999.9999 mm	0～999.99999 inch

Command execution process: as Fig. 2-50.

① The system executes the axial (Z) cutting feed △k from the starting point A_n of axial cutting cycle; when Z coordinate of cutting end point is less than that of starting point, the system executes Z negative feed, otherwise, positive feed;

② The system executes the axial(Z) rapid tool retraction e and its direction is opposite to the feed direction of ①;

③ The system executes Z cutting feed(Δk+e) again, the end point of cutting feed is still in it between starting point A_n of axial cutting cycle and end point B_n of axial tool infeed; the system executes Z cutting feed (Δk+e)again and then executes ②; after it executes Z cutting feed (Δk+e)again, the end point of cutting feed is on B_n or is not between A_n and B_n, the system executes Z cutting feed to B_n and then executes **4**;

④ Radial(X) rapid tool retraction △d（radius value）to C_n ; when X coordinate of B_f

(cutting end point) is less than that of A (starting point), the system executes X positive tool retraction, otherwise, X negative tool retraction;

⑤ Axial(Z axial) rapid retract tool to Dn, No. n axial cutting cycle is completed. If the current axial cutting cycle is not the last one, execute ⑥ ; if it is the previous one before the last axial cutting cycle, execute ⑦;

⑥ Radial(X axial)rapid tool infeed, and its direction is opposite to that of ④tool retraction. When the end point of tool infeed is still on it between A and A_f (starting point of last axial cutting cycle) after the system executes X tool infeed (△d+△i) (radius value) , i.e. Dn→A_{n+1} and then the system executes ① (start the next axial cutting cycle); after the system executes the tool infeed (△d+△i), the end point reaches A_f or is not between D_n and A_f, X rapidly traverse to A_f and executes ① to start the first axial cutting cycle;

⑦ X rapidly traverse to return to A, and G74 is completed.

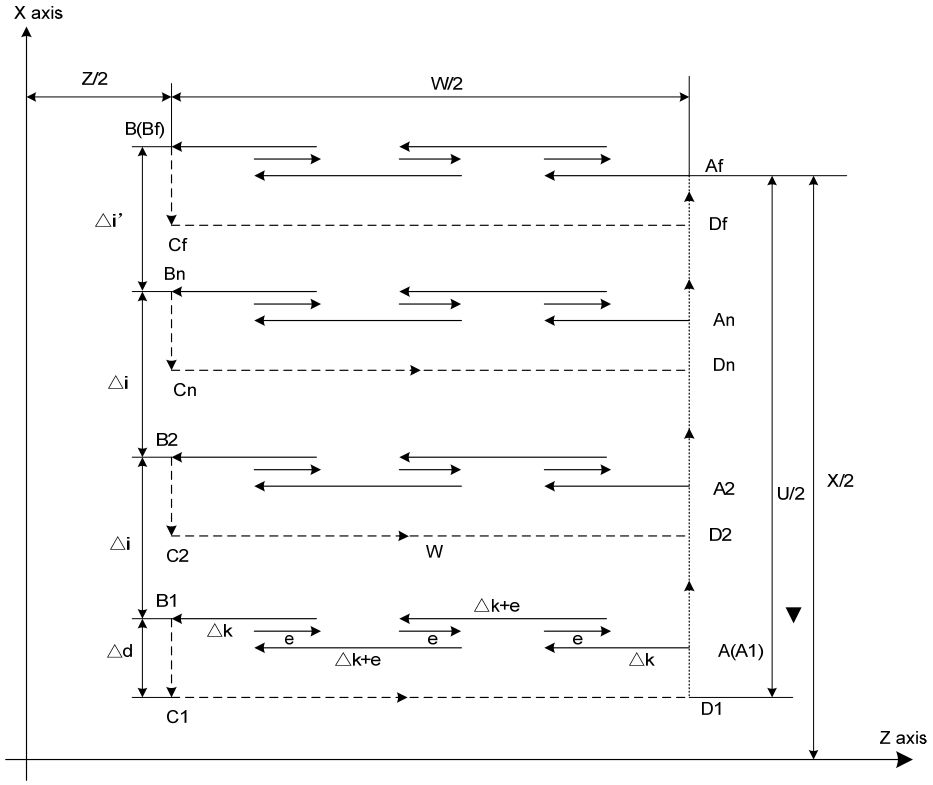


Fig. 2-50 G74 path

Example: Fig.2-51

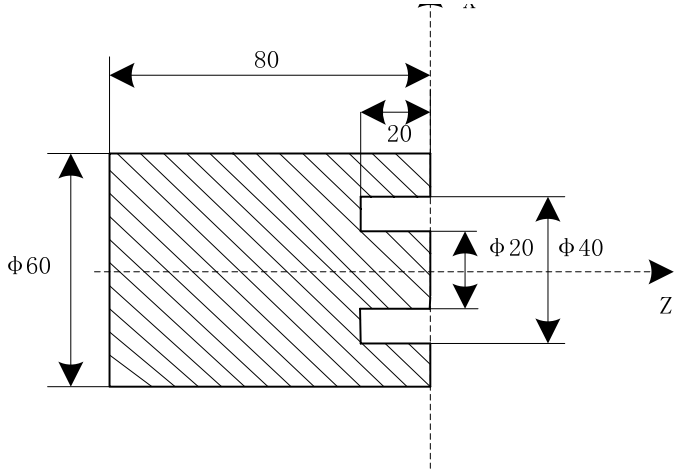


Fig. 2-51

Program：

O0007；

G0 X40 Z5 M3 S500； （Start spindle and position to starting point of machining）

G74 R0.5 ； （Machining cycle）

G74 X20 Z-20 P3000 Q5000 F50； (Z tool infeed 5mm and tool retraction 0.5mm each time; rapid return to starting point(Z5) after cutting feed to end point(Z-20), X tool infeed 3mm and cycle the above-mentioned steps)

M30； （End of program）

2.15.6 Radial Grooving Multiple Cycle G75

Command function: Axial (Z) tool infeed cycle compounds radial discontinuous cutting cycle: Tool infeeds from starting point in radial direction, retracts, infeeds again, and again and again, and last tool retracts in axial direction, and retracts to position in radial direction, which is called one radial cutting cycle; tool infeeds in axial direction and execute the next radial cutting cycle; cut to end point of cutting, and then return to starting point (starting point and end point are the same one in G75), which is called one radial grooving compound cycle. Directions of axial tool infeed and radial tool infeed are defined by relative position between end point X（U）Z（W） and starting point of cutting. G75 is used to machine the radial loop groove or column surface by radial discontinuously cutting, breaking stock and stock removal.

Command format： G75 R（e）；

G75 X（U）__ Z（W）__ P（Δi） Q（Δk） R（Δd） F__；

Command explanations:

1. The cycle movement is executed by X（W）and P（Δi）blocks of G75, G75 is not executed when there is no X(U) in G75 block. When only “G75 R（e）；” block is executed and only No.5139 value is modified, the cycle operation cannot be executed;
2. Δd and e are specified by the same address R and whether there are X（U）and P（Δi） words or not in blocks can distinguish them;

3. The tool can stop in Auto mode and traverse in Manual mode when G75 is executed, but the tool must return to the position before executing in Manual mode when G75 is executed again, otherwise the following path will be wrong;
4. When the system is executing the single block, the program pauses after the system has executed end point of current path;
5. R（Δd）must be omitted in grooving, and so there is no travel of tool retraction when the tool cuts to radial cutting end point.

Relevant definitions：

Starting point of radial cutting cycle	Starting position of axial tool infeed for each radial cutting cycle, defined by A_n(n=1,2,3……),X coordinate of A_n is the same that of starting point A, the different value of X coordinate between A_n and A_{n-1} is Δk. The starting point A_1 of the first radial cutting cycle is the same as the starting point A, and Z starting point (A_f) of the last axial cutting cycle is the same that of cutting end point.
End point of radial tool infeed	Starting position of radial tool infeed for each radial cutting cycle, defined by B_n(n=1,2,3……), X coordinates of B_n is the same that of cutting end point, Z coordinates of B_n is the same that of A_n , and the end point (B_f) of the last radial tool infeed is the same that of cutting end point.
End point of axial tool retraction	End position of axial tool infeed(travel of tool infeed is Δd) after each axial cutting cycle reaches the end point of axial tool infeed, defining with C_n(n=1,2,3……), X coordinate of C_n is the same that of cutting end point, and the different value of Z coordinate between C_n and A_n is Δd.
End point of radial cutting cycle	End position of radial tool retraction from the end point of axial tool retraction, defined by D_n(n=1,2,3……), X coordinate of D_n is the same that of starting point, Z coordinates of D_n is the same that of C_n (the different value of Z coordinate between it and A_n is Δd).
Cutting end point	It is defined by X（U）__ Z（W）___ ,and is defined with B_f of the last radial tool infeed.
R（e）	It is the travel of tool retraction after each radial(X) tool infeed without sign symbols and its value range is referred to the following table. The command value is reserved and the value of system parameter NO.5139 is rewritten after R（e）is executed. The value of NO.5139 is regarded as the travel of tool retraction when R（e）is not input.
X	X absolute coordinate value of cutting end point B_f (unit: mm).
U	Different value of X absolute coordinate between cutting end point B_f and starting point.
Z	Z absolute coordinate value of cutting end point B_f (unit: mm).
W	Different value of Z absolute coordinate between cutting end point B_f and starting point.
P（Δi）	It is the travel(diameter value) of radial(X) discontinuous tool infeed for each axial cutting cycle without sign symbols and its value range is referred to the following table.
Q（Δk）	It is the travel of Z discontinuous tool infeed without sign symbols of the axial(Z) cutting, and the value range is referred to the following table.
R（Δd）	It is the travel of axial (Z) tool retraction after cutting to end point of radial

	cutting with sign symbols and its value range is referred to the following table. The system defaults the axial(Z) tool retraction is 0 when R（Δd）和Q（Δk） are omitted. The system defaults to be the negative tool retraction when Z(W) is omitted.

Address	Incremental system	Metric（mm）input	Inch (inch) input
P（Δi）	ISB system	0~99999999（unit:0.001mm）	0~99999999（unit:0.0001inch）
Q（Δk）	ISC system	0~99999999（unit:0.0001mm）	0~99999999（unit:0.00001inch）
R（e）	ISB system	0～99999.999mm	0～9999.9999 inch
R（Δd）	ISC system	0～9999.9999 mm	0～999.99999 inch

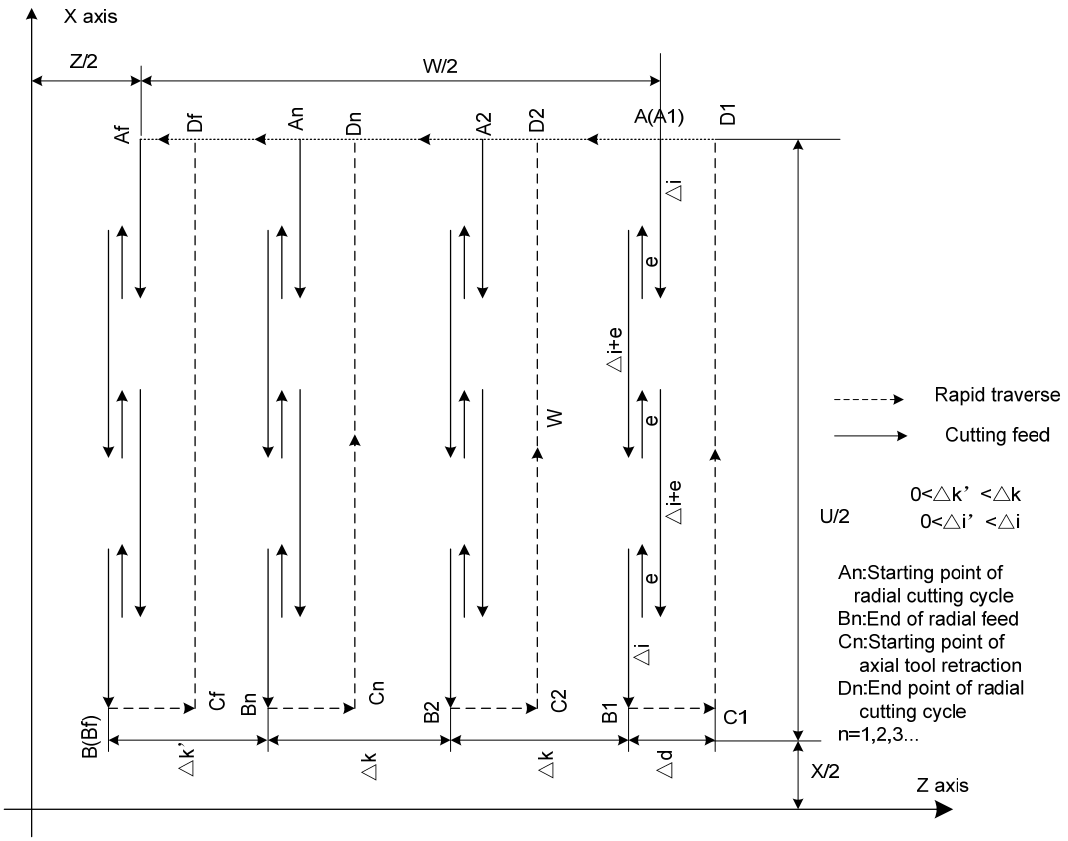


Fig. 2-52 G75 path

Execution process: as Fig. 2-52

① Radial (X) cutting feed △i from the starting point A_n of radial cutting cycle, feed in X negative direction when the coordinates of cutting end point is less than that of starting point in X direction, otherwise, feed in X positive direction;

② Radial(X) rapid tool retraction e and its direction is opposite to the feed direction of ①;

③ X executes the cutting feed (Δk+e) again, the end point of cutting feed is still in it between starting point A_n of radial cutting cycle and end point of radial tool infeed,

X executes the cutting feed (Δi+e)again and executes ②; after X cutting feed (Δi+e)is executed again, the end point of X cutting feed is on B_n or is not on it between A_n and B_n cutting feed to B_n and then execute ④;

④ Axial(Z) rapid tool retraction △d（radius value）to C_n , when Z coordinate of B_f (cutting end point) is less than that of A (starting point), retract tool in Z positive, otherwise, retract tool in Z negative direction;

⑤ Radial(X) rapid retract tool to Dn, No. n radial cutting cycle is completed. The current radial cutting cycle is not the last one, execute ⑥ ; if it is the previous one before the last radial cutting cycle, execute ⑦;

⑥ Axial(X)rapid tool infeed, and it direction is opposite to ④ retract tool. If the end point of tool infeed is still on it between A and A_f (starting point of last radial cutting cycle) after Z tool infeed (△d+△k) (radius value), i.e. Dn→A_{n+1} and then execute ① (start the next radial cutting cycle); if the end point of tool infeed is not on it between D_n and A_f after Z tool infeed (△d+△k) , rapidly traverse to A_f and execute ① to start the first radial cutting cycle;

⑦ Z rapidly moves to point A, G75 execution is completed.

Example: Fig.2-53

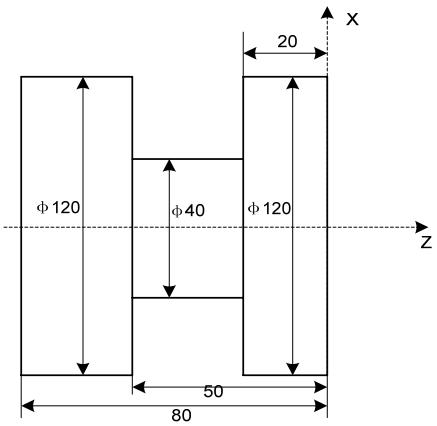


Fig.2-53 G75 cutting

Program：

O0008；

G00 X150 Z50 M3 S500； （Start spindle with 500 rev/min）

G0 X125 Z-20； （Position to starting point of machining）

G75 R0.5 F150； （Machining cycle）

G75 X40 Z-50 P6000 Q3000； （X tool infeed 6mm every time, tool retraction 0.5mm, rapid returning to starting point (X125) after infeeding to end point (X40), Z tool infeed 3mm and cycle the above-mentioned steps to continuously run programs）

G0 X150 Z50； （Return to starting point of machining）

M30； （End of program）

2.15.7 Notes for multi cycle machining

Note 1. When the multi cycle blocks are executed, they should be the specified address P, Q, X, Z, U, W, R of each block.

Note 2. The block specified by P in G71，G72, G73 should be G00G01. When there is no command, the system alarms.

Note 3. In MDI and DNC mode, G70，G71，G72 or G73 can not be specified, otherwise, the system alarms. But in MDI and DNC mode, G74，G75 or G76 can be specified.

Note 4: The block quantity of G70, G71, G72 or G73 in the sequence numbers specified by P and Q cannot exceed 100.

Note 5: The blocks in the serial numbers specified by P and Q in G71，G72 or G73 cannot specify the following command:

（1）non-modal G command except for G04 in group 00;

（2）all G commands except for G00, G01, G02, G03 in group 01;

（3）G20 and G21;

（4）M98 and M99;

Note 6: The skip function should not be executed in the blocks of their serial number specified by P and Q. when the skip function is used in the blocks of their serial numbers specified by P and Q.

Note 7: The tool nose radius compensation is invalid.

Note 8: No.5104 Bit2 (FCK）sets whether G71, G72, G73 executes the outer check. When it is set to1, the check is executed. The system alarms when the positioning point is in the cutting range.

Note 9: No.5102 Bit1（MRC）set whether the system alarm when the finishing cycle in G71，G72 is in non-monotonous, and it alarms when Bit1 is set to 1.

2.16 Threading Cutting

GSK988T CNC system can machine many kinds of thread cutting, including metric/inch single, multi threads, thread with variable lead and tapping cycle. Length and angle of thread run-out can be changed, multiple cycle thread is machined by single sided to protect tool and improve smooth finish of its surface. Thread cutting includes: continuous thread cutting G32, thread cutting with variable lead G34, Z thread cutting G33, Thread cutting cycle G92, Multiple thread cutting cycle G76

The machine used to thread cutting must be installed with spindle encoder, the transmission ratio between spindle and encoder is set by the parameter. X or Z traverses to start machine after the system receives spindle signal per rev in thread cutting, and so one thread is machined by multiple roughing, finishing without changing spindle speed.

GSK988T CNC system can machine many kinds of thread cutting, such as thread cutting without tool retraction groove. There is a big error in the thread pitch because there are the acceleration and the deceleration at the starting and ending of X and Z thread cutting, and so there is length of thread lead-in and distance of tool retraction at the actual starting and ending of thread cutting.

X, Z traverse speeds are defined by spindle speed instead of cutting feedrate override in thread cutting when the pitch is defined. The spindle override control is valid in thread cutting. When the spindle speed is changed, there is error in pitch caused by X and Z acceleration/deceleration, and so the spindle speed cannot be changed and the spindle cannot be stopped in thread cutting, which will cause tool and workpiece to be damaged.

Note: When the modal function is used, M30 (M30 modal in MDI mode cannot be cancelled) in Auto mode or G codes in Group 01 can cancel the mode state.

2.16.1 Thread Cutting with Constant Lead G32

Command function: Executing G32 can machine the metric or inch straight, taper, end face thread and continuous multi-section thread:

Command format：G32 X(U)_ Z(W)_ F(I)_ J_ K_ Q_

Command specifications: G32 is modal;

IP_	End point coordinate value. It can be specified by the absolute command value or incremental command value. The system specifies the different IP_ value to execute the straight thread cutting, end face thread cutting and taper thread cutting.
F	Metric pitch is moving distance of long axis when the spindle rotates one-turn and its value range is referred to the following table. After F is executed, it is valid until F with specified pitch is executed again. The pitch F value precision is the last two-digit of the decimal.
J	Travel in the short axis in thread run-out with positive/negative sign symbols and the value range is referred to the following table; the value is specified by the radius value.
K	Length in the long axis in thread run-out. The value range is referred to the following table. It has no direction.
Q	Initial angle between spindle rotation one-turn and starting point of thread cutting. The value range without the decimal is referred to the following table. Q is non-modal parameter, must be defined every time, it is 0^0 .when it is not specified, the system specifies Q different value can cut multi-thread.

Q rules:

1. Its initial angle is 0° if Q is not specified;
2. For continuous thread cutting, Q specified by its following thread cutting block except for the first block is invalid, namely Q is omitted even if it is specified;
3. In ISB mode, Q unit is 0.001°. ISC mode, Q unit is 0.0001°. Example, in ISB mode, Q180000 is input in program if it offsets 180° with spindle one rev; if Q180 or Q180.0, it is 0.18°. When the system specifies the value more than 360000, it counts based on 360000(180).
4. It is suggested that the system should use G97 instead of the constant surface cutting speed control in thread cutting.

Address	Incremental system	Metric（mm）input	Inch (inch) input
F	ISB, ISC	0.01～500 mm	0.01～9.99inch
J	ISB	-99999.999～99999.999mm	-9999.9999～9999.9999 inch
	ISC	-9999.9999～9999.9999 mm	-999.99999～999.99999 inch
K	ISB	0～99999.999mm	0～9999.9999 inch
	ISC	0～9999.9999mm	0～999.99999 inch
Q	ISB	0~99999999(unit:0.001 degree)	0~99999999（unit: 0.001 degree）
	ISC	0~99999999(unit:0.0001 degree)	0~99999999（unit:0.0001 degree）

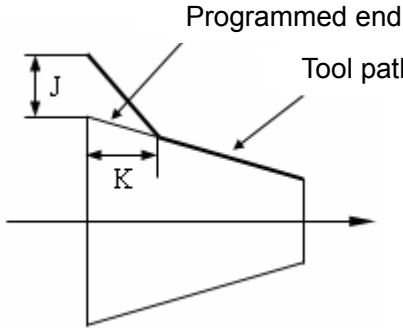


Fig. 2-54 thread run-out

Command path:

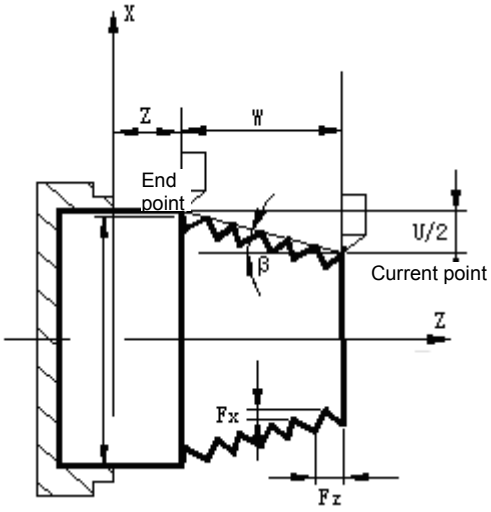


Fig.2-55 G32 path

Difference between long axis and short axis:

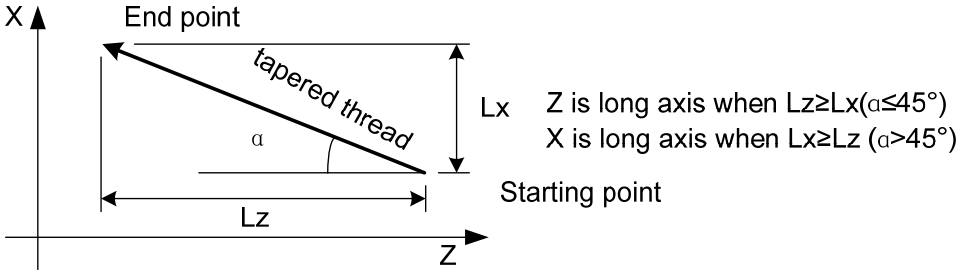


Fig.2-56 long axis, short axis

Note 1: When the thread run-out, the short axis executes the thread run-out at the rapid speed, and the long does it at the current thread cutting speed.

Note 2: J, K are modal. The thread run-out is previous J, K value when they are omitted in the next block in continuous thread cutting. Their mode are cancelled when no thread cutting are executed;

Note 3: There is no thread run-out when J, or J, K are omitted; K=J is the thread run-out value when K is omitted;

Note 4: There is no thread run-out when J=0 or J=0, K=0;

Note 5: The thread run-out value J=K when J≠0,K=0;

Note 6: There is no thread run-out when J=0 or K≠0;

Note 7: If the current block is for thread and the next block is the same, the system does not test the spindle encoder signal per rev at starting the next block to execute the direct thread cutting, which function is called as continuous thread machining;

Note 8: After the feed hold is executed, the system displays "Pause" and the thread cutting continuously

executes not to stop until the current block is executed completely; if the continuous thread cutting is executed, the program run pauses after thread cutting blocks are executed completely;

Note 9: In Single block, the program stops run after the current block is executed. The program stops run after all blocks for thread cutting are executed;

Note 10: The thread cutting decelerates to stop when the system resets, emergently stop or its driver alarms;

Note 11: The system alarms when the thread run-out length is more than the thread machined length of the long axis.

Note 12: In G32, the basic axis command cannot be in the same block with its parallel axis command, otherwise, the system alarms.

Note 13: When machining the thread in the metric tool machine in the unit of tooth/inch, using the expression calculated value programs F command. For example, when the thread with 10 teeth/inch is machined, using F[25.4/10] programs.

Note 14: The system automatically checks the spindle speed before machining the thread, the system alarms when the spindle speed is not commanded. The spindle speed cannot be checked in the course of the machining.

Example: Pitch: 2mm. δ1 = 3mm,δ2 = 2mm,total cutting depth 2mm with two times cut-in.

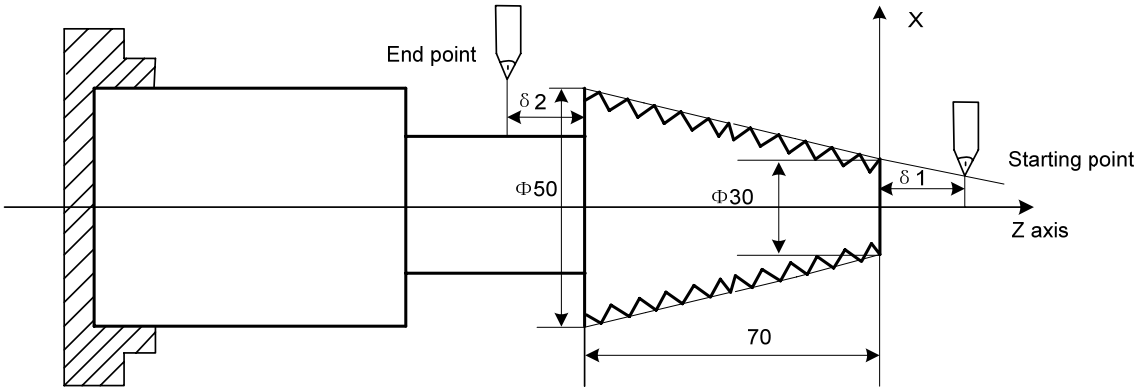


Fig. 2-57

Program:

```
O0009;
G00 X28 Z3;          (First cut-in 1mm)
G32 X51 W-75 F2.0;   (First taper cutting)
G00 X55;             (Tool retraction)
W75;                 (Z returns to the starting point)
X27;                 (Second tool infeed 0.5mm)
G32 X50 W-75 F2.0;   (Second taper thread cutting )
G00 X55;             (Tool retraction)
W75 ;                (Z returns to the starting point)
M30;
```

2.16.2 Thread cutting with variable lead G34

Command function: G34 can machine the metric, inch pitch. Machine metric or inch straight, taper, end face thread with variable pitch.

Command format: G34 X(U) __ Z(W) __ F(I) __ J__ K__ R__ ;

Command specifications: G34 is modal;

IP_, J_, K_, Q_	Meaning and value range are the same those of G32

F	It is the first thread pitch from starting point, and its range is the same that of G32
R	Incremental value or decremental value of spindle per pitch, R=F2-F1, R is with a direction; F1>F2, the pitch decreases when R is negative; F1<F2, the pitch increases when R is positive; R range: ±0.01 inch/pitch~ ±499.99 mm/pitch(metric thread); ±0.01 inch/pitch~ ±9.98 inch/pitch（inch thread）; An alarm occurs when R value exceeds the above range and the pitch exceeds the permissive range because of R increment/decrement or the pitch is negative.

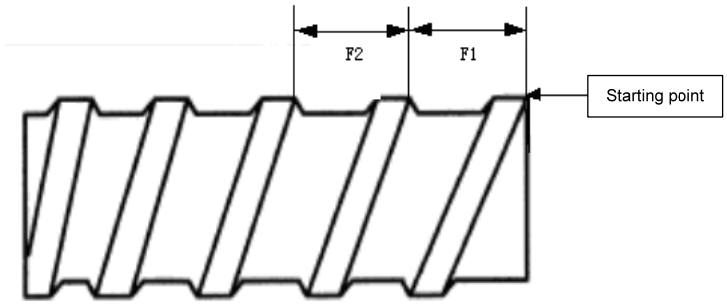


Fig. 2-58

Caution:

- It is the same as that of G32.

Example: First pitch of starting point**:** 4mm, increment 0.2mm per rev of spindle.

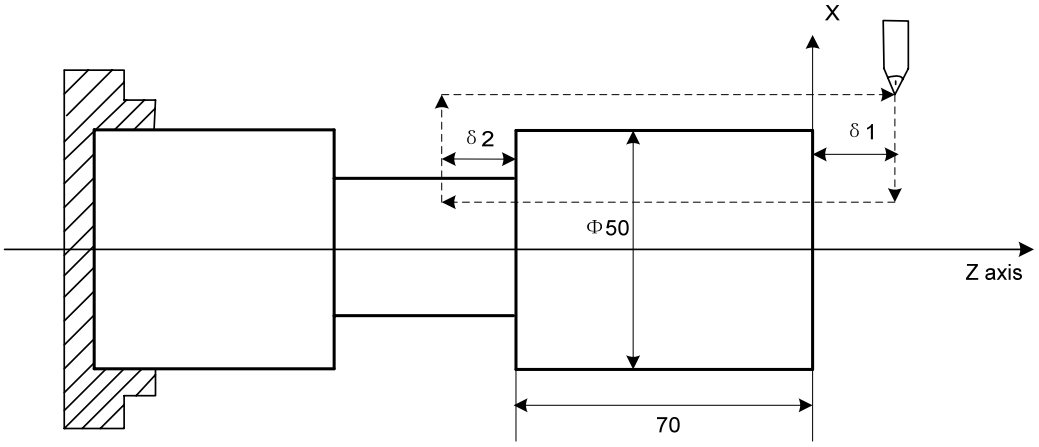


Fig. 2-59 Variable pitch thread machining

Value：δ1 = 4mm，δ2 = 4mm，total cutting depth 1mm，total cutting cycle 2 times；1st tool infeed 0.7mm.

Program：O0010；

```
G00 X60 Z4 M03 S500；
G00 U-10；                    Tool infeed Φ50
G00 U-0.7；                   Tool infeed
G34 W-78 F4 J5 K2 R0.2；      Variable pitch thread cutting
G00 U10；                     Tool retraction
Z4；                          Z returns to initial point
G00 X50；                     Tool infeed again Φ50
                              Tool infeed
```

```
G00 U-1.0；                    Variable pitch thread cutting
G34 W-78 F4 J5 K2 R0.2；       Tool retraction
G00 U10；                      Z returns to initial point
Z4；
M30；
```

2.16.3 Thread cutting cycle G92

Command function: Tool infeeds in radial(X) direction and cuts in axial(Z or X, Z) direction from starting point of cutting to realize straight thread, taper thread cutting cycle with constant thread pitch. Thread run-out in G92: at the fixed distance from end point of thread cutting, Z executes thread interpolation and X retracts with exponential or linear acceleration, and X retracts at rapidly traverse speed after Z reaches to end point of cutting as Fig. 2-52 and Fig.2-53.

Command format: G92 X（U）_ Z（W）_ F_ J_ K_ L_ Q_; （straight thread cutting cycle）

G92 X（U）_ Z（W）_ R_ F_ J_ K_ L_ Q_; （taper thread cutting cycle）

Command specifications: G92 is modal;

X	X absolute coordinate of end point of cutting
U	Different value of X absolute coordinate from end point to starting point of cutting
Z	Z absolute coordinate of end point of cutting
W	Different value of X absolute coordinate from end point to starting point of cutting
R	Different value(R value) of X absolute coordinate from end point to starting point of cutting. When the sign of R is not the same that of U, R｜≤｜U/2｜
F	Metric thread pitch is the same that of G32. After F value is executed, it is reserved and can be omitted
J	Travel in the short axis in thread run-out is same that of G32 and cannot be less than 0 without direction (automatically define its direction according to starting position of program), and it is modal parameter. If the short axis is X, its value is specified by radius
K	Travel in the long axis in thread run-out is same that of G32 without direction (automatically define its direction according to starting position of program), and it is modal parameter. If the long axis is X, its value is specified by radius
L	Multi threads: 1～99 and it is modal parameter. (The system defaults it is single thread when L is omitted)
Q	Shift angle of initial angle at the beginning of thread cutting. Its range is referred to G32.

The system can machine one thread with many tool infeed in G92, but cannot do continuous two thread and end face thread. Definition of thread pitch in G92 is the same that of G32, and a pitch is defined that it is a moving distance of long axis(X in radius) when the spindle rotates one revolution.

Pitch of taper thread is defined that it is a moving distance of long axis(X in radius). When absolute value of Z coordinate difference between B point and C point is more than that of X (in

radius), Z is long axis; and reversely, X is the long axis.

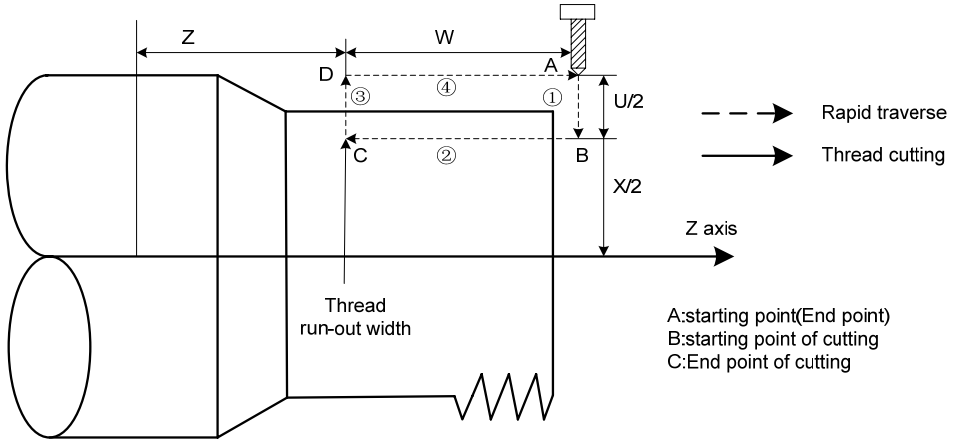


Fig. 2-60 Straight thread

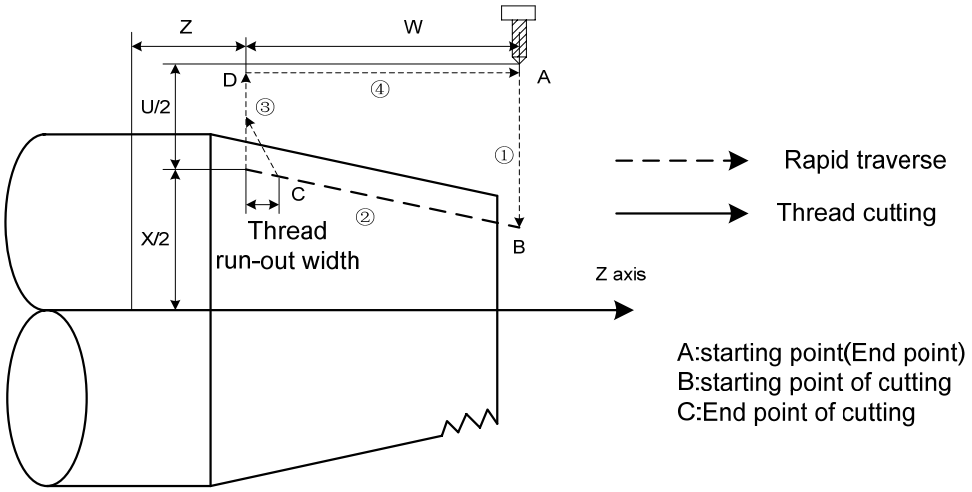


Fig.2-61 Taper thread

Cycle process: straight thread as Fig.2-60 and taper thread as Fig.2-61.

① X traverses from starting point to cutting starting point;

② Thread interpolates (linear interpolation) from the cutting starting point to cutting end point;

③ X retracts the tool at the cutting feedrate (opposite direction to the above-mentioned ①), and return to the position which X absolute coordinate and the starting point are the same;

④ Z rapidly traverses to return to the starting point and the cycle is completed.

Note 1: When J, K is omitted, the thread run-out is confirmed by No. 5130 (chamfering value) and No. 1531 (run-out angle) , the run-out value of the long axis =No. 5130 setting value ×0.1×F, and F is the thread pitch. When No. 5131 (the run-out angle) is set to 0, the long axis and the short axis execute 45°run-out; when the setting value is positive integer, the run-out is done based on the run-out value and angle of the long axis(the system automatically counts the run-out value of the short axis);

Note 2: Length of thread run-out is K in the long direction and is specified by No.5130 when J is omitted;
Note 3: Length of thread run-out is J=K when K is omitted;
Note 4: There is no thread run-out when J=0 or J=0, K=0;
Note 5: Length of thread run-out is J=K when J≠0,K=0;
Note 6: There is no thread run-out when J=0,K≠0;
Note 7: After executing the feed hold in thread cutting, the system does not stop cutting until the thread

cutting is completed with *Pause* on screen;

Note 8: After executing single block in thread cutting, the program run stops after the system returns to starting point(one thread cutting cycle is completed);

Note 9: Thread cutting decelerates to stop when the system resets, emergently stops or its driver alarms;

Note 10: The system alarms when the thread run-out length of the long axis is more than the thread machining length of the long axis;

Note 11: The system alarms when the thread run-out length of the short axis is more than the thread machining length of the short axis;

Note 12: The system automatically checks the spindle speed, and an alarm occurs when the spindle speed is not specified. The spindle speed cannot be checked during the machining.

Example:

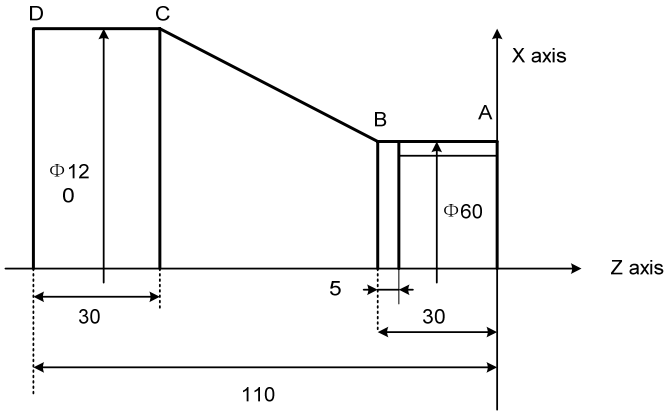


Fig. 2-62

Program:

```
O0012;
M3 S300 G0 X150 Z50 T0101;     (Thread tool)
G0 X65 Z5;                      (Rapid traverse)
G92 X58.7 Z-28 F3 J3 K1;        (Machine thread with 4 times cutting, the first tool
                                 infeed 1.3mm)
X57.7 ;                        (The second tool infeed 1mm)
X57;                            (The third tool infeed 0.7mm)
X56.9;                          (The fourth tool infeed 0.1mm)
M30;
```

2.16.4 Multiple thread cutting cycle G76

Command function: Machining thread with specified depth of thread (total cutting depth)is completed by multiple roughing and finishing, if the defined angle of thread is not 0°, thread run-in path of roughing is from its top to bottom, and angle of neighboring thread teeth is the defined angle of thread. G76 can be used for machining the straight and taper thread with thread run-out path, which is contributed to thread cutting with single tool edge to reduce the wear of tool and to improve the precision of machining thread. But G76 cannot be used for machining the face thread. machining path is as Fig.2-55.

Command format：G76 P（m）（r）（a） Q（△dmin） R（d）；

G76 X（U）__ Z（W）__ R（i） P（k） Q（△d） F_ J_ K_；

Command explanations:

Starting point (end point)	Position before block runs and behind blocks run, defined by A point.
End point of thread	End point of thread cutting defined by X(U) __ Z(W) __,.The tool will not reach the point in cutting if there is the thread run-out path.
Starting point of thread	Its absolute coordinates is the same that of A point and the different value of X absolute coordinates between C and D is i(thread taper with radius value). The tool cannot reach C point in cutting when the defined angle of thread is not 0°.
Reference position of thread cutting depth	Its absolute coordinates is the same that of A point and the different value of X absolute coordinate between B and C is k(thread taper with radius value).The cutting depth of thread at B point is 0 which is the reference position used for counting each thread cutting depth by the system.
Thread cutting depth	It is the cutting depth for each thread cutting cycle. It is the different value (radius value, without signs) of X absolute coordinate between B and intersection of reversal extension line for each thread cutting path and straight line BC. The cutting depth for each roughing is $\sqrt{n}\times\triangle d$, n is the current roughing cycle times, $\triangle d$ is the thread cutting depth of first roughing.
Travel of thread cutting	Different value between the current thread current depth and the previous one:$(\sqrt{n}-\sqrt{n-1})\times\triangle d$
End point of tool retraction	It is the end position of radial (X) tool retraction after the thread cutting in each thread roughing, finishing cycle is completed, is defined by E point. $$tg\frac{a}{2}=\frac{|Z\ \text{replacement}|}{|X\ \text{replacement}|}$$ a：thread angle
Thread cut-in point	Actual start thread cutting point in each thread roughing cycle and finishing cycle. It is defined by (n is the cutting cycle times), B_n is the first thread roughing cut-in point, $B_1$is the last thread roughing cut-in point, B_e is the thread finishing cut-in point. B_n is the X, Z displacement formula for B point.
X	X absolute coordinate of thread end point.
U	Difference value of X absolute coordinate between thread end point and starting point.
Z	Z absolute coordinate of thread end point.
W	Different value of Z absolute coordinate between thread end point and starting point.
P(m)	Times of thread finishing: 00～99 (unit: times) with 2-digit digital. It is valid after m command value is executed, and the value of system parameter No.5142 is rewritten to m. The value of system parameter No.5142 is regarded as finishing times when m is not input. The thread is finished according to the programmed thread path, the first finishing cutting travel is d and the following one is 0.

P(r)	Width of thread run-out 00～99(unit: 0.1×L,L is the thread pitch) with 2-digit digital. It is valid after r command value is executed and the value of system parameter No.5130 is rewritten to r. The value of system parameter No.5130 is the width of thread run-out when r is not input. The thread run-out function can be applied to thread machining without tool retraction groove and the width of thread run-out defined by system parameter No.5130 is valid for G92, G76.
P(a)	Angle at taper of neighboring two tooth is 0~99, unit: degree(°) ,with 2-digit digital. It is valid after a command value is executed and the value of system parameter No.5143 is rewritten to a. The value of system parameter №058 is regarded as angle of thread tooth. The actual angle of thread in defined by tool ones and so a should be the same as the tool angle.
△Q(△dmin)	Minimum cutting travel of thread roughing (radius value without sign symbols). When ($\sqrt{n}-\sqrt{n-1}$) ×△d＜△dmin, △dmin is regarded as the cutting travel of current roughing, i.e. depth of current thread cutting is ($\sqrt{n-1}$ ×△d+△dmin) . △dmin is applied because the cutting travel of roughing is undersize and the times of roughing is excessive, which is caused the cutting travel of thread roughing gradually decreases. After Q(△dmin) is executed, the command value △dmin is value and the value of system parameter No.5140 is rewritten to minimum cutting travel; when Q（△dmin） is not input, the system takes No.5140 value as the least cutting value.
R(d)	It is the cutting travel of thread finishing, and is the different value(radius value without sign symbols) of X absolute coordinates between cut-in point Be of thread finishing and Bf of thread roughing. After R(d) is executed, the command value d is value and the value of system parameter No.5141 is rewritten to d×1000(unit: 0.001 mm) . The value of system parameter No.5141 is regarded as the cutting travel of thread finishing when R(d) is not input.
R(i)	It is thread taper and is the different value of X absolute coordinate between thread starting point and end point (unit: mm, radius value). The system defaults i=0(straight thread) when i is not input.
P(k)	It is the depth of thread tooth and is also the total cutting depth of thread(radius value without sign symbols), and the system alarms when P(k) is not input.
Q(△d)	It is the first depth of thread cutting (radius value without sign symbols).The system alarms when △d is not input.
F	Pitch is defined to moving distance (radius value in X direction) of long axis when the spindle rotates one rev. Z is long when absolute value of coordinate difference between C point and D point in Z direction is more than that of X direction (radius value, be equal to absolute value of i); and vice versa.
J	When the thread run-out is executed, the movement range in the short axis direction is the same that of G32, must not be less than 0 without direction

	(the system automatically confirms the run-out direction according to the initial pont of the program), is modal and its value is specified by radius.
K	When the thread run-out is executed, the range in the long axis direction is the same that of G32, is modal without direction, and the value is specified by radius.

Address	Incremental system	Metric （mm）input	Inch (inch) input
Q(△dmin)	ISB system	0~99999999（unit：0.001mm）	0~99999999（unit：0.0001inch）
	ISC system	0~99999999（unit：0.0001mm）	0~99999999（unit：0.00001inch）
R（d）	ISB system	0.001～99999.999 mm	0.0001～9999.9999 inch
	ISC system	0.0001～9999.9999 mm	0.00001～999.99999 inch
R（i）	ISB system	-99999.999～99999.999 mm	-9999.9999～9999.9999 inch
	ISC system	-9999.9999～9999.9999mm	-999.99999～999.99999 inch
P(k)	ISB system	1~99999999（unit：0.001mm）	1~99999999（unit：0.0001inch）
	ISC system	1~99999999（unit：0.0001mm）	1~99999999（unit：0.00001inch）
Q(△d)	ISB system	1~99999999（unit：0.001mm）	1~99999999（unit：0.0001inch）
	ISC system	1~99999999（unit：0.0001mm）	1~99999999（unit：0.00001inch）
F	ISB,ISC	0.01～500 mm	0.01～9.99inch

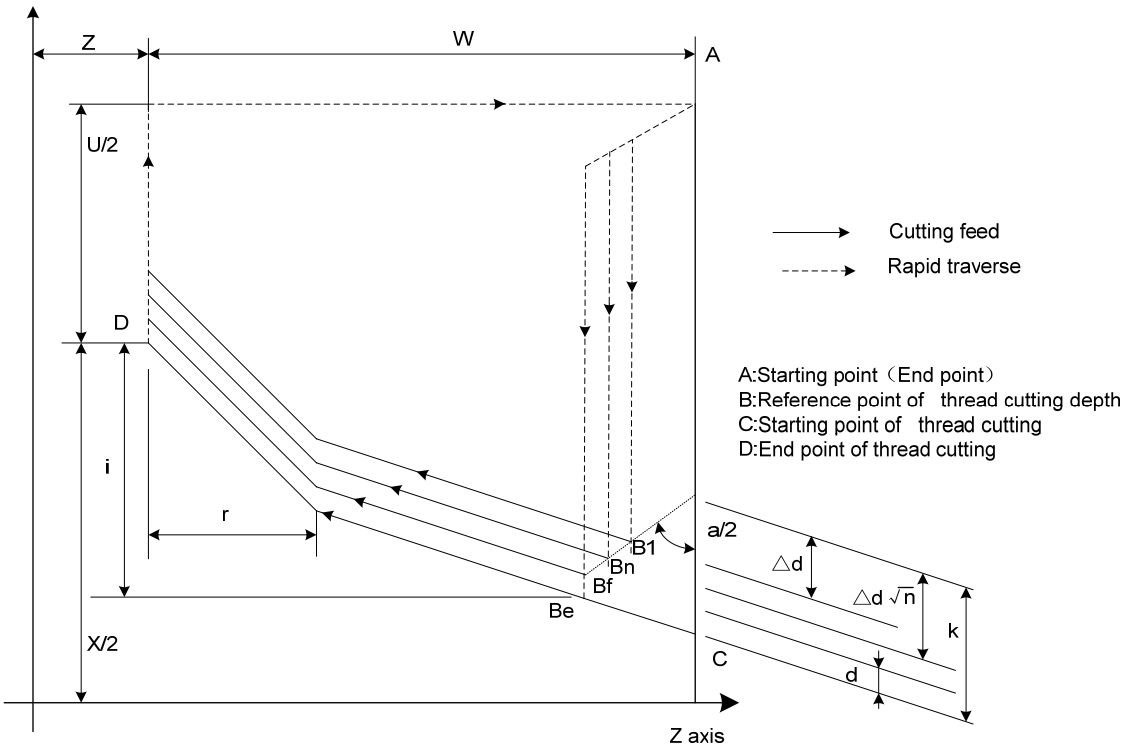


Fig.2-63

Cut-in method is shown in Fig. 2-64:

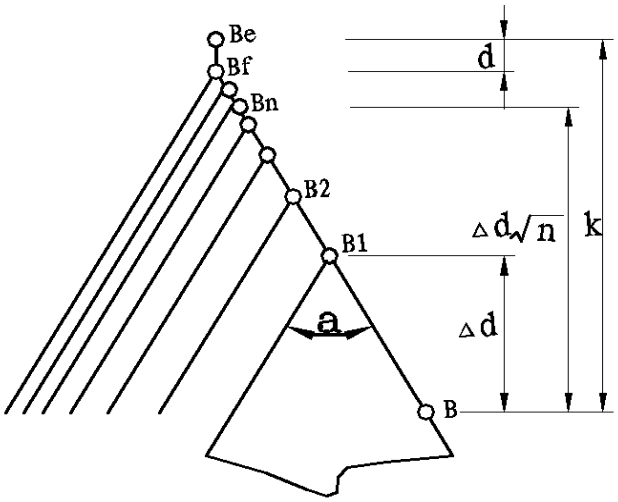


Fig. 2-64

Execution process:

① The tool rapidly traverses to B_1, and the thread cutting depth is △d. The tool only traverses in X direction when a=0; the tool traverses in X and Z direction and its direction is the same that of A→D when a≠0;

② The tool cuts threads paralleling with C→D to the intersection of D→E (r≠0: thread run-out);

③ The tool rapidly traverses to E point in X direction;

④ The tool rapidly traverses to A point in Z direction and the single roughing cycle is completed;

⑤ The tool rapidly traverses again to tool infeed to B_n (is the roughing times), the cutting depth is the bigger value of ($\sqrt{n}$ ×△d) ,($\sqrt{n-1}$ ×△d+$\triangle d_{min}$) , and execute ② if the cutting depth is less than(k-d) ; if the cutting depth is more than or equal to(k-d) , the tool infeeds (k-d) to B_f, and then execute ⑥ to complete the last thread roughing;

⑥ The tool cuts threads paralleling with C→D to the intersection of D→E (r≠0: thread run-out);

⑦ X axis rapidly traverses to E point;

⑧ Z axis traverses to A point and the thread roughing cycle is completed to execute the finishing;

⑨ After the tool rapidly traverses to B(the cutting depth is k and the cutting travel is d), execute the thread finishing, at last the tool returns to A point and so the thread finishing cycle is completed;

⑩ If the finishing cycle time is less than m, execute ⑨ to perform the finishing cycle, the thread cutting depth is k and the cutting travel is 0; if the finishing cycle times is equal to m, G76 compound thread machining cycle is completed.

Note 1: When G76 is executed, after【FEED HOLD】is pressed and the system executes this thread cutting cycle, the system enters pauses state and "Stop" appears in the status column;

Note 2: The single block is executed during the course of thread cutting, the run stops after the system returns to starting point (one thread cutting cycle is completed);

Note 3: The thread cutting speed stops when the system resets, emergently stops or the drive unti alarms;

Note 4: All or some addresses of G76 P（m)(r)(a) Q(△dmin) R(d) are omitted, and omitted addresses runs according to the setting value;

Note 5: m, r, a uses the same address P to be input one time. When m, r, a are all omitted,the system runs at the setting value of No.5142, No.5130 or No.5143; when P is with non regular value, the system takes the last two digits of P value as a value, and the last threeth and fourth digits as r value, and the left as m value;

Note 6: Signs of U, W determines direction of A→C→D→E, R (i) determines that of C→D. Four kind of combincation of U, W correspond to 4 kind of machining path;

Note 7: When the set first thread cutting depth is more than the total cutting depth, one roughing is executed, and its cutting depth is equal to the total cutting depth of roughing;

Note 8: When the least cutting amount or finishing allowance is more than thread tooth height in the course of thread roughing, an alarm occurs;

Note 9: When the run-out length is more than the machining thread length of long axis, an alarm occurs;

Note 10: The system automatically checks the spindle speed before machining, an alarm occurs when the spindle speed is not commanded. The spindle speed cannot be checked in the course of machining;

Note 11：The run-out format is the same that of G32 when there is J, K.

Example: Fig. 2-65, thread M68×6.

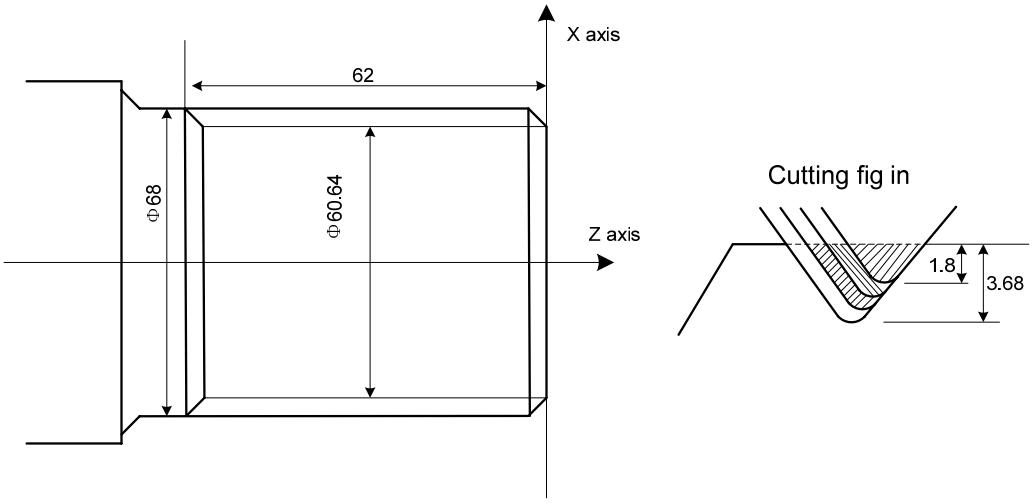


Fig.2-65

Program:

```
O0013;
G50 X100 Z50 M3 S300;          (Set workpiece coordinate system, start spindle
                                and specify spindle speed)
G00 X80 Z10;                   (Rapid traverse to starting point of machining)
G76 P020560 Q150 R0.1;         (Finishing 2 times, chamfering width 0.5mm,
                                tool angle 60°, min. cutting depth 0.15,
                                finishing allowance 0.1)
```

```
G76 X60.64 Z-62 P3680 Q1800 F6;   (Tooth height 3.68, the first cutting depth 1.8)
G00 X100 Z50 ;                    (Return to starting point of program)
M30;                              (End of program)
```


2.17 Constant Surface Speed Control G96, Constant Rotational Speed Control G97

Command function: The constant surface speed control is valid, the cutting surface speed is defined (m/min) and the constant rotational speed control is cancelled.

Command format: G96 Sxxxx;

Command explanation: G96 is modal G command. If the current modal is G96, G96 can not be input; it is the cutting surface speed in Sxxxxx constant surface control.

Command function: the constant surface speed control is cancelled, the constant rotational speed control is valid and the spindle speed is defined (r/min).

Command format: G97 Sxxxx;

Command explanation: G96 is modal G command. If the current modal is G97, G97 cannot be input; it is the spindle speed in Sxxxxx constant speed control(r/min).

Relative command: G50

Command function: define max. spindle speed limit (r/min) in the constant surface speed control(r/min).

Command format: G50 Sxxxx;

Command explanation: After the system is turned on, and the max. spindle speed is not specified, the system does not limit the spindle speed state. Max. spindle speed limit is valid for G96, and is invalid for G97;

S value set by G50 is modal and is value before the new max. speed is set;

Note: when G50 S0 is executed, the spindle speed is limited in 0 r/min (the spindle does not rotate) in the constant surface control;

The system does not set the current workpiece coordinate system when G50 sets the constant surface speed control.

Address	Incremental system	Metric （mm）input	Inch (inch)input
S（G96）	ISB, ISC	0～20000 m/min	0～2000 feet/min
S（G97）	ISB, ISC	0～20000 r/min	0～20000 r/min

When the machine tool cuts it, the workpiece rotates based on the axes of spindle as the center line, the cutting point of tool cutting workpiece is a circle motion around the axes of spindle, and the instantaneous speed in the circle tangent direction is called the cutting surface(for short surface speed). There are different surface speed for the different workpiece and tool with different material.

When the spindle speed controlled by the analog voltage is valid, the constant surface control is valid. The spindle speed is changed along with the absolute value of X absolute coordinate of programming path in the constant speed control. If the absolute value of X absolute coordinate increases, the spindle speed reduces, and vice versa, which make the cutting surface speed as S command value. The constant speed control to cut the workpiece makes sure all smooth finish on

the surface of workpiece with diameter changing.

Surface speed=spindle speed× |X| × π ÷1000 (m/min)

Spindle speed: r/min

|X|: absolute value of X absolute coordinate value (diameter value), mm

π≈3.14

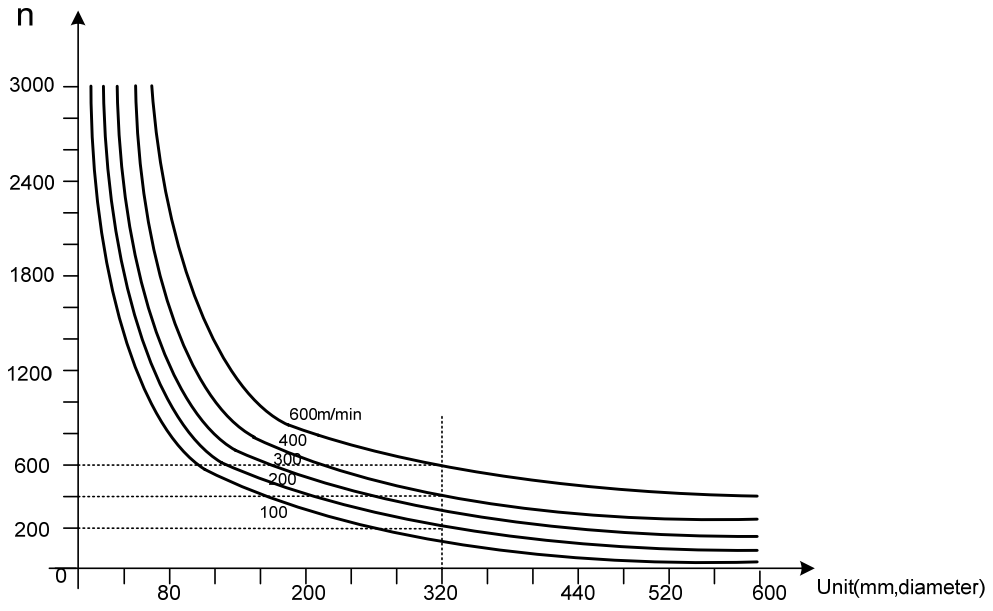


Fig. 2-66

In G96, the spindle speed is changed along with the absolute value of programming path X absolute coordinate value in the course of cutting feed (interpolation), but it is not changed in G00 because there is no actual cutting and is counted based on the surface speed of end point in the program block.

In G96, Z coordinates axis of workpiece system must consist with the axes of spindle (rotary axis of workpiece), otherwise, there is different between the actual surface speed and the defined one.

When the constant surface speed is valid, G50 S_ can limit max. spindle speed (r/min). The actual spindle speed is the limit value of max. speed when the spindle speed counted by the surface speed and X coordinate value is more than the max. spindle speed set by G50 S_. After the system powers on, max. spindle speed limit value is not defined and its function is invalid. Max. spindle speed limit value defined by G50 S_ is reserved before it is defined again and its function is valid in G96. Max. spindle speed defined by G50 S_ is invalid in G97 but its limit value is reserved.

Note 1: G96, G97 are modal in the same group, and one of them is valid in the same time. G97 is initial word and is valid after the system is turned on.

Note 2: In G96, S value commanded is reserved in G97. there is no new S is commanded and the S value in the last G96 state is recovered to the current valid surface speed after the system returns to G96 state, the system outputs the least surface speed in G96 when there is no saved value.

Note 3: From G96 to G97, if none of S command (r/min) is commanded in the program block in G97, the last spindle speed in G96 is taken as S command in G97, namely, the spindle speed is not changed at this time;

Note 4: The constant surface speed control function is still valid when the machine is locked(X, Z do not move when the system executes X, Z motion commands);

Note 5: In G96, when the spindle speed counted by the cutting surface speed is more than max. speed of

current spindle gear, at this time, the spindle speed is limited to max. one of current spindle gear;

Note 6: In thread cutting, To gain the precise thread machining, it should not be adopted with the constant surface speed control but the constant rotational speed (G97) in the course of thread cutting;

Note 7: No.3031 sets the numerical digit permitted by S.

Note 8: X=0: the theory speed is infinite but the actual speed corresponds to 10V voltage because the maximum voltage of sent analog is 10V.

Example:

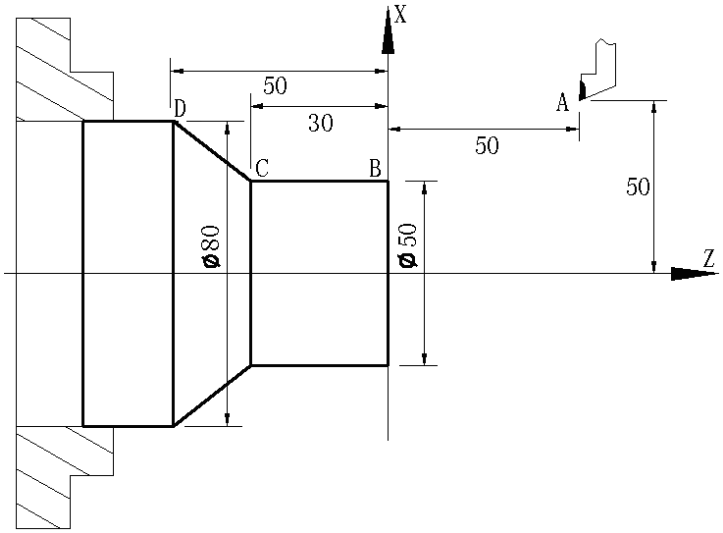


Fig.2-67

Program：

M3 G96 S300;	(Spindle rotates clockwise, the constant surface speed control is valid and the surface speed is 300m/min)
G0 X100 Z100;	(Rapid traverse to A point with spindle speed 955 r/min)
G0 X50 Z0;	(Rapid traverse to B point with spindle speed 1910 r/min)
G1 W-30 F200;	(Cut from B to C with spindle speed 1910 r/min)
X80 W-20 F150;	(Cut from C to D with spindle speed 1910 r/min and surface speed 1194 r/min)
G0 X100 Z100;	(Rapid retract to A point with spindle speed 955 r/min)
M30;	(End of program, spindle stop and cooling OFF)

2.18 Feedrate per Minute G98, Feedrate per Rev G99

Command function: Cutting feed rate is specified as mm/min, G98 is the modal G command. G98 cannot be input if the current command is G98 modal.

Command format: G98 Fxxxx; (F0001～NO027, the leading zero can be omitted, feed rate per minute is specified, mm/min)

Command function: Cutting feed rate is specified as mm/min, G99 is the modal G command. G99 input may be omitted if current state is G99.

Command format: G99 Fxxxx; (F0.0001～F500, the leading zero can be omitted)

Command explanation:

When G99 Fxxxx is executed, the actual cutting feedrate is gotten by multiplying the F command value (mm/r) to the current spindle

speed(r/min). If the spindle speed varies, the actual feedrate changes too. If the spindle cutting feed amount per rev is specified by G99 FXXXX , the even cutting texture on the surface of workpiece will be gotten. In G99 state, a spindle encoder should be fixed on the machine tool to machine the workpiece.

F range in G98, G99 is shown below.

Address	Incremental system	Metric（mm）input	Inch (inch)input
F（G98）	ISB system	1～60000 mm/min	0.01～2400 inch/min
	ISC system	1～24000 mm/min	0.01～960 inch/min
F（G99）	ISB system	0.01～500mm/r	0.01～9.99inch/r
	ISC system	0.01～500mm/r	0.01～9.99 inch/r

Reduction formula of feed between per rev and per min:

$$F_m = F_r \times S$$

F_m: feed per min (mm/min) ;

F_r: feed per rev(mm/r) ;

S: spindle speed (r/min) .

F value is reserved after the system executes F command.

Note 1: G98, G99 are the modal G commands in the same group and only one is valid. G98 is the initial state G command and the system defaults the modal can be set by No.3402 Bit4 (FPM) when the system turns on.

Note 2: In G99 modal, there is the uneven cutting feed rate when the spindle speed is lower than 1 r/min; there is the follow error in the actual cutting feed rate when there is the swing in the spindle speed. To gain the high machining quality, it is recommended that the selected spindle speed should be not lower than min. speed of spindle servo or converter.

Note 3: No.1422 set the upper of the cutting feedrate. When the actual cutting feedrate (the value is multiplied by the override) exceeds the specified upper limit, it is clamped to the upper limit value.

Note 4: No. 1403 Bit0(MIF)can set the cutting speed unit per minute and the detailed is referred to the parameter explanation.

Note 5: When G99 instead of F command in G98 mode is commanded, F is the previous modal value in G99. In a similar way, when G98 instead of F command in G99 mode is commanded, F is the previous modal value in G98.

Note 6: When the initial mode is G98/99, and G99/G98 is alone executed after power on, the system runs at the speed set by No. 1411.

2.19 Drilling/Boring Fixed Cycle Command

Many blocks completes one machining in the course of drilling. To simplify programming, GSK988T uses one drilling cycle G commands to complete a series of drilling machining. (C tool compensation vector in the course of drilling/boring will temporarily cancel, automatically recovers after the command is completed)

- Execution process

  The drilling fixed cycle is composed of the following 6 operations.

  Operation 1: X(Z) and C axis (requirement in some occasion) positions to the hole positon of initial level;

  Operation 2: rapidly traverse to point R;

  Operation 3: drilling (cutting feed or interval feed);

  Operation 4: pause at the hole bottom;

  Operation 5: retract tool to the level where point R is;

Operation 6: rapidly traverse to initial level

Operation sequence

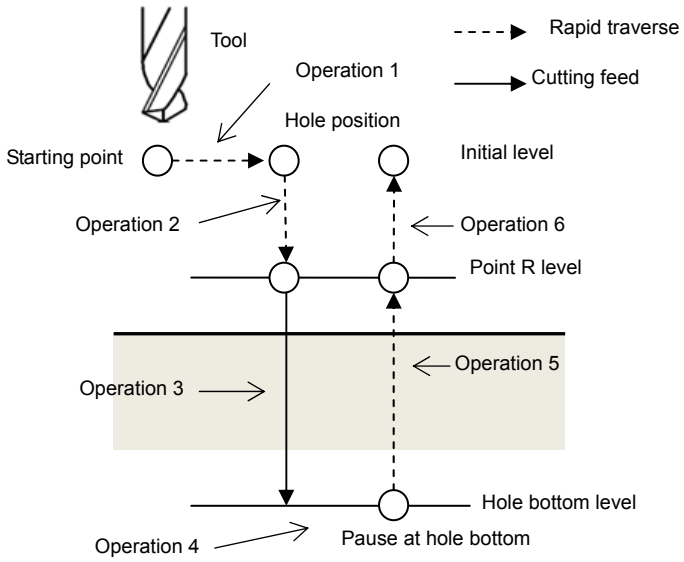


Fig. 2-68

- Drilling fixed cycle G commands（included in Group 10）

G command	Drilling axis	Drilling operation	Operation at bottom	Tool retraction	Function
G83	Z	Interval feed/ cutting feed	Pause	Rapid traverse	End drilling cycle
G87	X	Interval feed / cutting feed	Pause	Rapid traverse	Side drilling cycle
G85	Z	Cutting feed	Pause	Cutting feed	End boring cycle
G89	X	Cutting feed	Pause	Cutting feed	Side boring cycle
G80	/	/	/	/	Cancel drilling fixed cycle

- Positioning axis and drilling axis
  G confirms the drilling axis and the positioning axis is the others except for the drilling axis.

G command	Drilling axis	Positioning axis
G83, G85	Z	X and C
G87, G89	X	Z and C

Note: C axis can be omitted.

- Fixed cycle is cancelled
  G80 or G commands included in Group 01 can cancel the fixed cycle.

2.19.1 End drilling cycle G83 /side drilling cycle G87

Command format：G83 X(U)_ C(H)_ Z(W)_ R_ P_ Q_ F_ K_ M_；or

G87 Z(W)_ C(H)_ X(U)_ R_ P_ Q_ F_ K_ M_；

Command definition:

X_ C_ or Z_ C_	It is hole position data, and valid in the specified block.
Z(W)_ or X(U)_	The absolute value specifies the coordinates of hole bottom

	or the incremental value specifies the distance from R level to the hole bottom, which is value in the specified block.
R_	It is the distance from the initial level to point R, is specified by radius value with direction. Its unit and range are shown in the following table.
P_	It is pause time at the bottom. ISB system unit is 1ms，ISC system unit is 0.1ms.
Q_	It is cutting amount every time and specified by radius value. Cutting amount, radius value every time, unit and range are shown in the following table.
F_	Cutting feedrate.
K_	Program execution times.
M_	M command for clamping C axis（it is used when C is needed to clamp）.

+	Incremental system	Metric input （mm）	Inch input (inch)
Q	ISB system	0~99999999（unit：0.001mm）	0~99999999（unit：0.0001inch）
	ISC system	0~99999999（unit：0.0001mm）	0~99999999（unit：0.00001inch）
R	ISB system	-99999.999～99999.999mm	-9999.9999～9999.9999 inch
	ISC system	-9999.9999～9999.9999 mm	-999.99999～999.99999 inch
K	ISB system	1～99 times	1～99 times
	ISC system	1～99 times	1～99 times

In G83/87, high speed deep hole drilling cycle, deep hole drilling cycle and standard drilling cycle can be selected by Q value (cutting amount every time) and RTR (NO.5101#2).

High speed deep hole drilling cycle	Q value is specified（Q value is not zero） and the parameter RTR（NO.5101#2）=“0”
Deep hole drilling cycle	Q value is specified（Q value is not zero） and the parameter RTR（NO.5101#2）=“1”
Standard drilling cycle	Q value is not specified or Q value is zero.

G83, G87 are modal, remain valid once are specified until the fixed cycle is cancelled.

- **High speed deep hole drilling cycle**（Q value is specified (it is not zero) and RTR（NO.5101#2）=“0”）

  The system executes the intermittent cutting and chip removal with the specified tool retraction amount before entering the hole bottom, which is executed repetitively until the tool infeeds to the bottom, and then the tool retraction is performed, so the machining is completed.

Command format and definition are referred to the previous description.

Execution process:

① The tool rapidly positions to the hole from starting point （the hole is determined by the hole position data at the initial level）;
② Rapidly position to point R;
③ Cutting feed executes the cutting amount q specified by Q;
④ Rapid tool retraction executes retraction amount d specified by No. 5114;
⑤ Repeat the above Step ③④ until the tool reaches the level where the hole bottom is;
⑥Pause is executed in the time specified by P;
⑦ Return rapidly to the level where point R is;
⑧ Return rapidly to the initial level;
⑨ Drilling cycle ends.

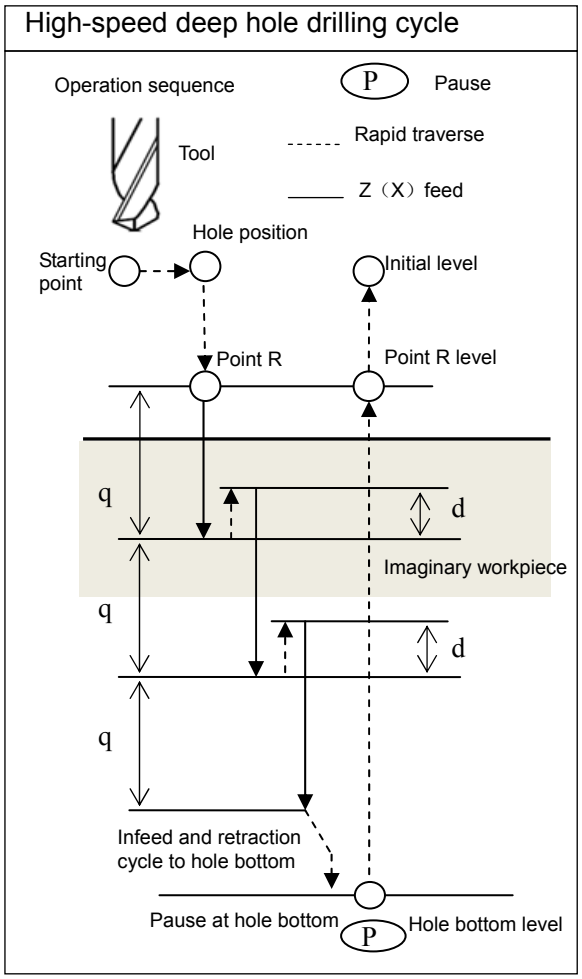


Fig.2-69

- **Deep hold drilling cycle**（Q value is specified and RTR（NO.5101#2）="1"）

  Command format and definition are referred to the previous description.

Execution process:

① The tool rapidly positions to the hole from starting point（the hole is determined by the hole position data at the initial level）;
② Rapidly position to point R;
③ Cutting feed executes the cutting amount q specified by Q;
④ Rapidly retract to the level where point R is;
⑤ Rapid feed to the position which is d from the previous machining level (NO.5115 specifies the dry running amount d of deep hole drilling cycle）;
⑥ The cutting feed (the distance q+d) is executed;
⑦Repeat the above Step ③④ until the tool reaches the level where the hole bottom is;
⑧ Pause is executed in the time specified by P;
⑨ Return rapidly to the level where point R is;
⑩ Return rapidly to the initial level;
⑪ Drilling cycle ends.

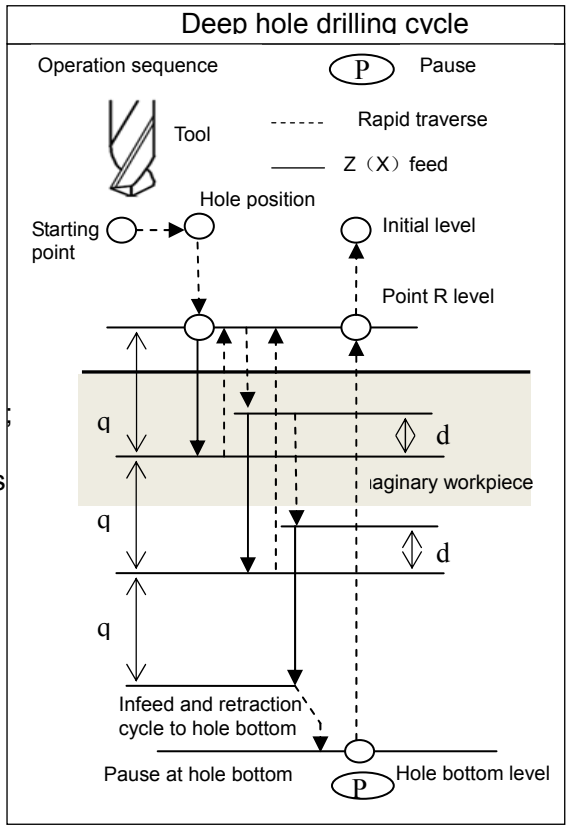


Fig.2-70

- **Standard drilling cycle**（Q value is not specified）

Command format：G83 X(U)_ C(H)_ Z(W)_ R_ P_ F_ K_ M_; or

G87 Z(W)_ C(H)_ X(U)_ R_ P_ F_ K_ M_;

Command explanation：the command definition is referred to the previous description.

Execution process:

① The tool rapidly positions to the hole from starting point (the hole is determined by the hole position data at the initial level）;
② Rapidly position to point R;
③ The cutting feed is executed to the hole bottom;
④ Pause is executed in the time specified by P;
⑤ Rapidly retract to the level where point R is;
⑥ Return rapidly to the initial level;
⑦ Drilling cycle ends.

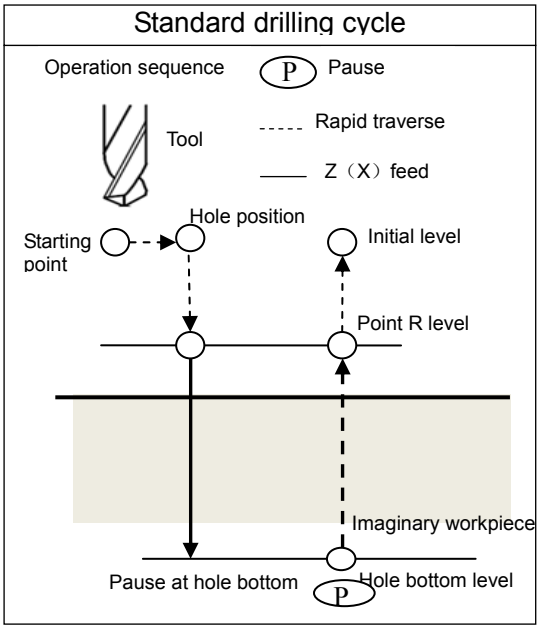


Fig. 2-71

Program example:

Suppose the current system is ISB, the minimum input unit is 0.001mm and RTR is set to 1.

```
G98                          ；feed mode per minute
M51                          ；activate C indexing（suppose M51 is for
                               activating C indexing）
M3 S1500                     ；tool starts rotation
G0 X50 C0 Z-4                ；X and C axis position to the starting point
G83 X100 Z-50 R4 Q5000 P3000 ；starting point is X50 C0，hole position is
F200                         X100 C0，
                             ；point R is X100 Z-4，hole position is X100
                             Z-50，
                             ；the cutting amount every time is 5mm,
                             pause time is 3s.
                             ；the block is for deep hole drilling according
                             to Q value and RTR
C120                         ；position to C120 to drilling the 2nd point
C240                         ；position to C240 to drilling the 3rd point
G80 M05                      ；the fixed cycle is cancelled, the tool stops
                               rotation
M50                          ；C axis indexing closes（suppose M50 is for
                               closing C axis indexing）
M30                          ；end of program
```

2.19.2 End Boring CycleG85 / Side Boring Cycle G89

The cycle is used for executing boring operation.

Command format：G85 X(U)_ C(H)_ Z(W)_ R_ P_ F_ K_ M_；or
G89 Z(W)_ C(H)_ X(U)_ R_ P_ F_ K_ M_；

Command definition：

X_ C_ or Z_ C_	It is the hole position data and is valid only in the specified block.
Z(W)_or X(U)_	It specifies the coordinate value of hole bottom by using absolute coordinate , or specifies the distance from R level to the hole bottom by using incremental value, and it is valid in the specified block.
R_	It is the distance from the initial level to point R and is specified by radius value with direction. Its unit and range is shown below.
P_	Hole bottom pause time. Unit of ISB system is 1ms and ISC is 0.1ms.
F_	Cutting feed speed.
K_	Execution times of program（it is used when it is needed）.
M_	M command for clamping C axis（it is used when it is needed）.

Relevant command explanation is referred to those of G83/87.

Execution process:

① The tool rapidly positions to the hole from starting point (the hole is determined by the hole position data at the initial level）;
② Rapidly position to point R;
③ The cutting feed is executed to the hole bottom at the speed specified by F;
④ Pause is executed in the time specified by P;
⑤ Rapidly retract to the level where point R is; (No.5149 is used for setting the override of boring retraction. When it is set to 0, the double speed of F value is default to execute tool retraction
⑥ Return rapidly to the initial level;
⑦ Drilling cycle ends.

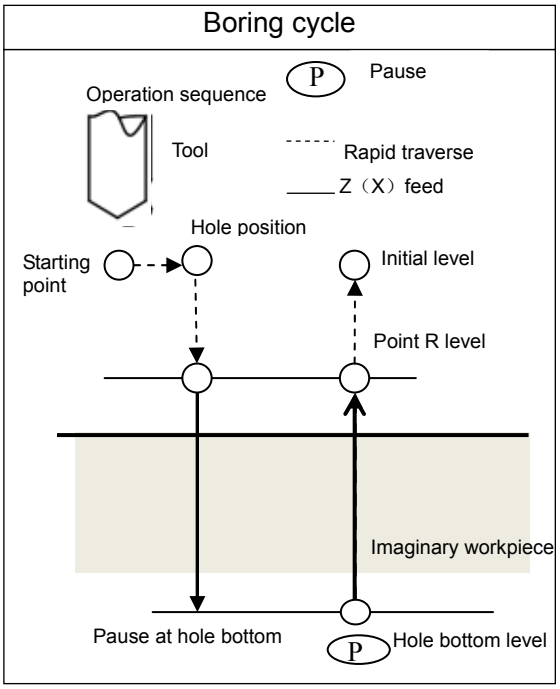


Fig. 2-72

2.19.3 Cancelling Drilling/Boring G80

The command is used for cancel the drilling fixed cycle.

Command format: G80;

After G80 is executed, the hole position data, R and other drilling data are cancelled, and the mode of drilling cycle is also done.

2.19.4 Notes for Drilling/Boring Cycle

Note 1: When the reset or emergency stop is executed, the mode of drilling cycle remains. The user must pay more attention to it when the program is started again.

Note 2: The single block stops at end point of operation 1, operation 2 or operation 6.

Note 3: When drilling/boring cycle is executed, creating or cancelling tool compensation command is executed, the command is valid after the cycle ends.

2.20 Tapping Cycle Command

GSK988T CNC Turning System uses end tapping cycle (G84) and side tapping cycle (G88) to complete the tapping function. Tapping is divided into common tapping (flexible) and rigid tapping mode. In the common tapping mode, the spindle rotation and feed amount of tapping axis are controlled separately, their synchronous relationship is not controlled well. In the rigid tapping mode, the control of spindle motor is the same that of servo motor, the spindle rotating one circle corresponds to some axial feed amount of the spindle even if the spindle accelerates/decelerates. In the rigid tapping, the spindle can rapidly and exactly tap without using the floating chuck or variable screw tap(use it in the common tapping mode).

M29(it can set other M command according to parameter or directly use G command to

specify rigid mode without M command)specifies the common tapping cycle and rigid tapping cycle when programming.

When the rigid tapping is executed, the machine must have the corresponding conditions, i.e. the spindle uses the position control and is applied to Cs axis, otherwise the system does not support the function. The function is applied to the machine with high configuration.

End tapping cycle (G84), side tapping cycle (G88), drilling fixed cycle G83/G87 and boring cycle G85/G89 are in the same Group 10. G80 or one command included in Group 01 can cancel the tapping fixed cycle. The system executes the normal operation after the drilling fixed cycle is cancelled. Clear point R and hole bottom (point X or Z) data and other tapping data (P, K, F) is also cleared.

Vector of C tool compensation during the course of tapping is temporarily cancelled, but automatically recovers after the command is executed.

2.20.1 Tapping Mode

Tapping cycle is divided into common mode and rigid tapping mode, and the follow method can specify the rigid tapping mode; when N0.5200#0=0 and M29 is not specified, the system executes the common tapping mode.

1） Specify M29 S**** before G84 (G88) blocks;

```
M29 S_;
G84 X_ C_ Z_ R _ P_ F_ K_ (M_);
X_ C_;
G80;
```

2） It is specified in the same block in G84 (G88) tapping blocks; M command for clamping C axis cannot be specified in G84/G88 blocks in the mode.

```
G84 X_ C_ Z_ R _ P_ F_ K_ M29 S_;
X_ C_;
G80;
```

3） G84/G88 is used for rigid tapping(Bit0 of No.5200 is set to 1); in the mode, G84/G88 is used for only the rigid tapping mode instead of the common tapping mode.

```
G84 X_ C_ Z_ R _ P_ F_ K_ M _;
X_ C_;
G80;
```

M29 (the parameter sets other M command to specify it) is for rigid tapping, the system alarms when S is specified between M29 and G84/G88 blocks or the axis movement command is specified; the system alarms when M39 is specified repetitively in tapping cycle (M29 cannot be specified repetitively).

M29 Sxxxx commands rigid tapping mode. The corresponding switch is done after PLC receives M29 and the spindle stops rotation. The spindle output is equivalent to S0 output in M29.

2.20.2 End Rigid Tapping Cycle (G84) / Side Rigid Tapping Cycle (G88)

Command function：When the spindle is controlled in rigid mode (it is taken as the servo motor), the rigid tapping cycle is executed.

Command format ： G84 X (U)_ C (H)_ Z (W)_ R_ Q_ P_ F_ K_ M_ ; or
G88 Z (W)_ C (H)_ X (U)_ R_ Q_ P_ F_ K_ M_ ;

Command explanation ：

X_ C_ or Z_ C_	It is the hole position data and is valid only in the specified block; the hole position data can specify other valid axes except for X, Z, C.
Z(W)_ or X(U)_	It specifies the coordinate value of hole bottom by using absolute coordinate, or specifies the distance from R level to the hole bottom by using incremental value, and it is valid in the specified block.
R_	It is the distance from the initial level to point R and is specified by radius value with direction. Its unit and range is shown below.
P_	Hole bottom pause time. Unit of ISB system is 1ms and ISC is 0.1ms.
Q_	Cutting amount every time is specified by radius value. Its unit and range are shown below. When Q value is specified, G84/G88 selects the high speed deep hole rigid tapping cycle or deep hole rigid tapping cycle by PCP (No. 500#5). Q value is not specified or Q value is 0, the standard rigid tapping cycle is selected.
F_	Cutting feed speed.
K_	Execution times of program（it is used when it is needed）.
M_	M command for clamping C axis（it is used when it is needed）.

+	Incremental system	Metric input（mm）	Inch input(inch)
Q	ISB system	0~99999999(unit：0.001mm)	0~99999999(unit: 0.0001inch)
	ISC system	0~99999999(unit：0.0001mm)	0~99999999(unit: 0.00001inc)）
R	ISB system	-99999.999～99999.999mm	-9999.9999～9999.9999 inch
	ISC system	-9999.9999 ～9999.9999 mm	-999.99999 ～999.99999 inch

Tapping feed axis specifies X or Z axis according to G84/G88. G84 specifies Z to be the tapping axis and G88 specifies X. The spindle is selected according to relevant G signals (it is related to PLC programs).

Cutting feedrate F (i.e. feedrate of tapping axis) and spindle speed S confirm the thread lead.

Thread lead formula in per minute mode =cutting feedrate F/spindle speed S;

Thread lead formula in per rotation mode=cutting feedrate F.

In rigid tapping mode, three machining modes (standard rigid tapping cycle, high speed deep hole rigid tapping cycle and deep hole rigid tapping cycle) are selected by Q value (cutting amount every time) and PCP (NO.5200#5) in GG84/88

Standard rigid tapping cycle	Q value is not specified or Q value is 0
High speed deep hole rigid tapping cycle	Q value is specified（it is not zero）and PCP（NO.5200#5）="0"
Deep hole rigid tapping cycle	Q value is specified（it is not zero）and RTR（NO.5200#5）="1"

- Standard rigid tapping cycle（Q value is not specified or Q value is 0）

  Command format： G84 X (U)_ C (H)_ Z (W)_ R_ P_ F_ K_ M_ ; or
  G88 Z (W)_ C (H)_ X (U)_ R_ P_ F_ K_ M_ ;

Execution process:

① The tool rapidly positions to the hole from starting point (the hole is determined by the hole position data at the initial level）;
② Rapidly position to point R;
③ The spindle starts rotation, and tapping axis is executed to the hole bottom level at the speed specified by F, and the spindle stops when the axis reaches the hole bottom;
④ Pause is executed in the time specified by P;
⑤ The spindle starts rotation reversely and tapping axis retracts to the R level at the speed specified by F;
⑥ The spindle stops rotation and return rapidly to the initial level;
⑦ The standard tapping cycle ends.

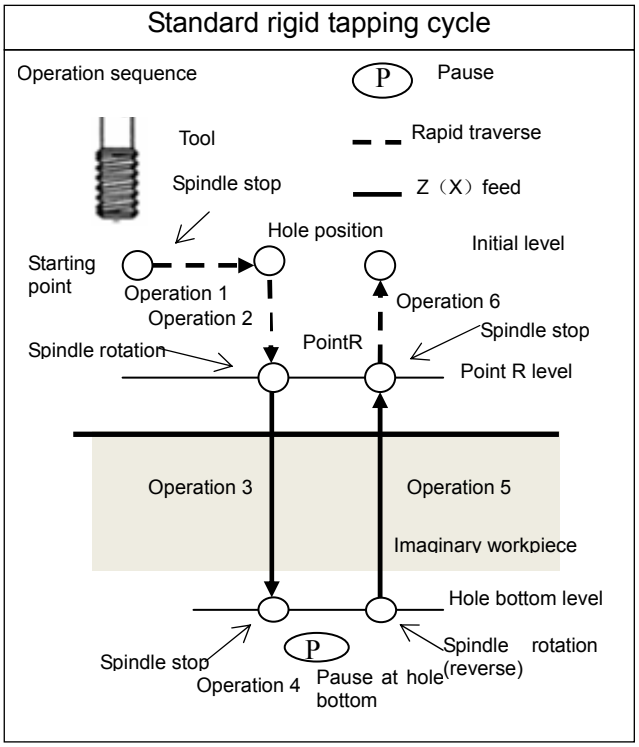


Fig 2-73

- High speed deep hole rigid tapping cycle (Q value is specified（it is not zero）and PCP（NO.5200#5）="0")

  Before the tool enters the hole bottom, the intermittent tapping is executed and the chip removal is done with the specified tool retraction amount, which are done repetitive until the tool reaches the hole bottom, then the tool retracts and the machining ends.

  Command format： G84 X (U)_ C (H)_ Z (W)_ R_ Q_ P_ F_ K_ M_ ; or
  G88 Z (W)_ C (H)_ X (U)_ R_ Q_ P_ F_ K_ M_ ;

Execution process:

① The tool positions the hole position from Starting point（i.e. the point on the initial level is confirmed by the hole position);
② Rapidly position to point R;
③ The spindle starts rotation;
④ The tapping axis feeds at the cutting speed F with the cutting amount q; the spindle stops after the feed ends;
⑤ The spindle rotates reversely, and the tapping axis executes the tool retraction amount d set by No.5213; the spindle stops after tool retraction ends;
⑥ Repeat the above ③④⑤ till the tool reaches The hole bottom level; the spindle stops;
⑦ Pause is executed in the time specified by P;
⑧ The spindle rotates reversely, and the tapping axis returns to point R level at the specified speed;
⑨ Rapidly return to the initial level;
⑩ Standard rigid tapping cycle ends.

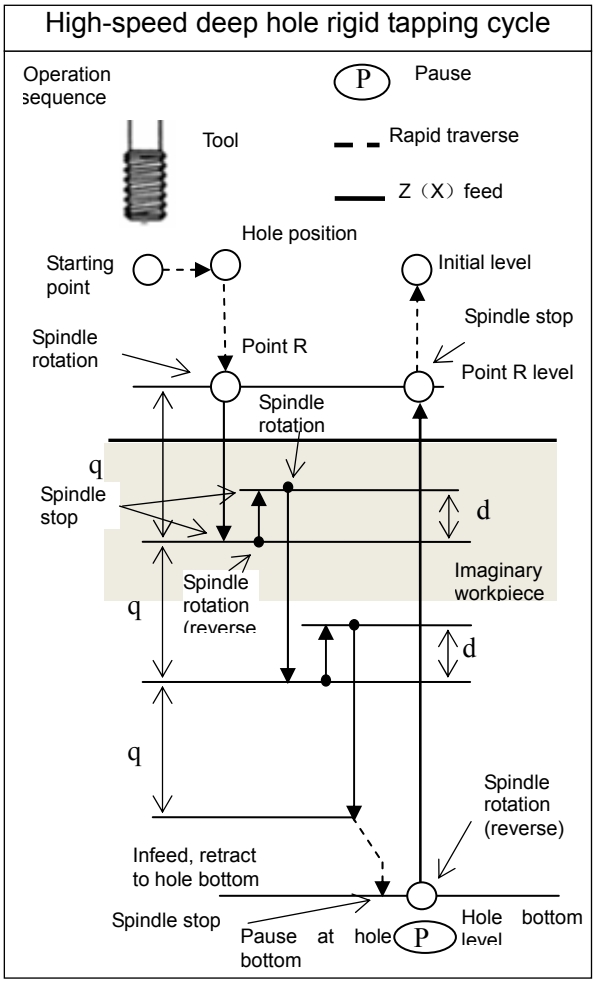


Fig 2-74

- Deep hole rigid tapping cycle（Q value is specified（it is not zero）and RTR（NO.5200#5）="1"）

  The cycle executes the deep hole rigid tapping operation.

  Command format： G84 X (U)_ C (H)_ Z (W)_ R_ Q_ P_ F_ K_ M_ ; or
  G88 Z (W)_ C (H)_ X (U)_ R_ Q_ P_ F_ K_ M_ ;

Execution process:

① The tool rapidly positions to the hole from starting point (the hole is determined by the hole position data at the initial level）;
② Rapidly position to point R;
③ The spindle starts rotation;
④ The tapping axis feeds at the cutting speed F with the cutting amount q specified by Q the spindle stops after the feed ends;
⑤ The spindle rotates reversely, and the spindle stops rotation after the tapping axis executes the tool retraction to point R level;
⑥ The spindle starts rotation, the tappind axis executes infeed at the previous machine level D (), and the retraction amount d) of deep tapping is specified by No.5213.
⑦ The cutting feed of tapping is q+d;
⑧ Repeat the above ⑤⑥⑦ till the tool reaches the hole bottom level; the spindle stops;
⑨ Pause is executed in the time specified by P;
⑩ The spindle rotates reversely, and the tapping axis returns to point R level at the specified speed;
⑪ Rapidly return to the inItial level;
⑫ Deep hole rigid tapping cycle ends.

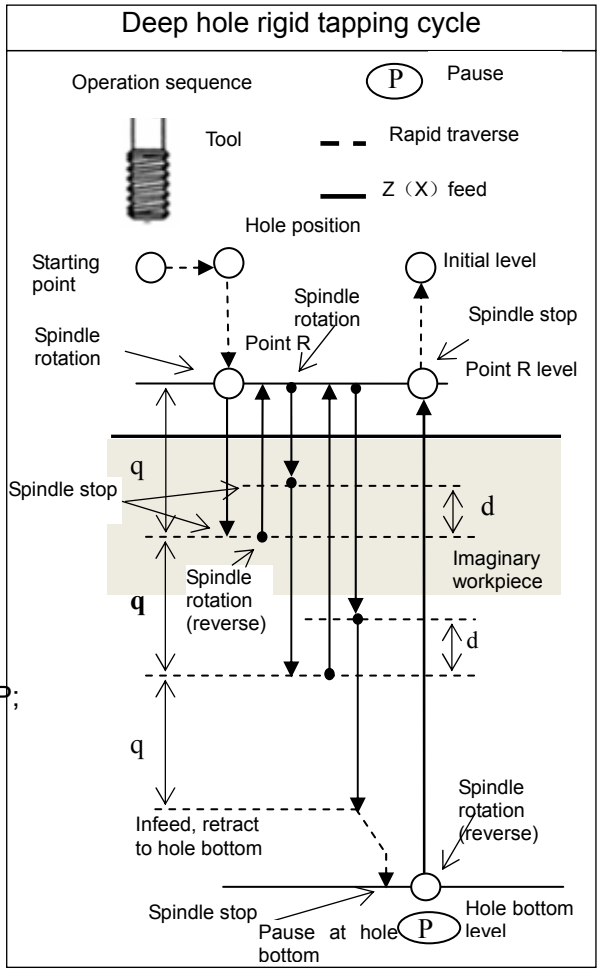


Fig. 2-75

Note 1: In the rigid tapping cycle, the speed of tool retraction and the one of the tool cutting to the previous machining level are specified by feedrate F (for tapping axis, the feedrate is the specified F in the command, which can distinguish F98/G99; for the spindle, the feedrate is the specified spindle speed) and extraction override.

The extraction override is fixed to 100% when the parameter DOV(No. 5200#4, whether the tool retraction is valid in the rigid tapping) is set to 0.
When DOV (No.5200#4) is set to 1, it is divided into the following conditions:

（1）The extraction override is set by No.5211(it is the override value in rigid tapping) , among which the parameter OVU(NO.5201#3) is used to set the setting unit of extraction override parameter in rigid tapping when the parameter OV3 (NO.5201#4 confirms the extraction override by the address specifying whether the spindle speed is valid). Namely, unit of No.5211 is 1% or 10%,

（2）When OV3 is set to 1, J address specifies the spindle speed in tool retraction.

$$\text{Tapping override}(\%) = \frac{\text{Spindle speed when tapping (J command)}}{\text{Spindle speed (S command)}} \times 100$$

Besides, when the override value is out of the range 100%～2000%, it becomes 100%. When the extraction is executed, the spindle speed address "J" is valid before the fixed tapping cycle is cancelled in rigid tapping mode.

OVE(No.5202#6)="0":

Spindle speed command when extraction is done		DOV= "1"		DOV= "0"
		OV3= "1"	OV3= "0"	
Spindle speed command with "J" specifying extraction:	Within 100～200%	Programmed command	(No.5211)	100%
	Out of 100～200%	100%		
Spindle speed command without "J" specifying extraction:		(No.5211)		

OVE(No.5202#6)="1":

Spindle speed command when extraction is done		DOV= "1"		DOV= "0"
		OV3= "1"	OV3= "0"	
Spindle speed command with "J" specifying extraction:	Within 100～2000%	Programmed command	(No.5211)	100%
	Out of 100～2000%	100%		
Spindle speed command without "J" specifying extraction:		(No.5211)		

Note 2: Specify P/Q in the blocks for drilling operation. It is not taken as the modal data to store when it is specified in the blocks not for drilling operation.

The deep hole rigid tapping operation is not executed when Q0 is specified.

Note 3: Retraction amount d is set to the value (No. 5213) which does not exceed the cutting amount q when the deep hole tapping cycle is executed.

Note 4: R is the distance from the initial level to point R and is specified by radius value, and the initial level is considered to R level after it is omitted.

Note 5: G84/88 is used for dry run. Feedrate F is the one in Dry run mode.

Note 6: For feed pause, single block, when G84/G88 fixed cycle is at the operation 1, 2 and 6, "Feed Pause" is pressed to decelerate; when it is at the operation 3, 4, 5 (tapping), the movement does not immediately stop until the tool returns to the level where point R is. When G84/88 is executed in single block mode or the single block mode is opened in the cycle, the single block stops run at the end point of the operation 1, 2, 6 (operation 3, 4, 5, 6 are combined into one block).

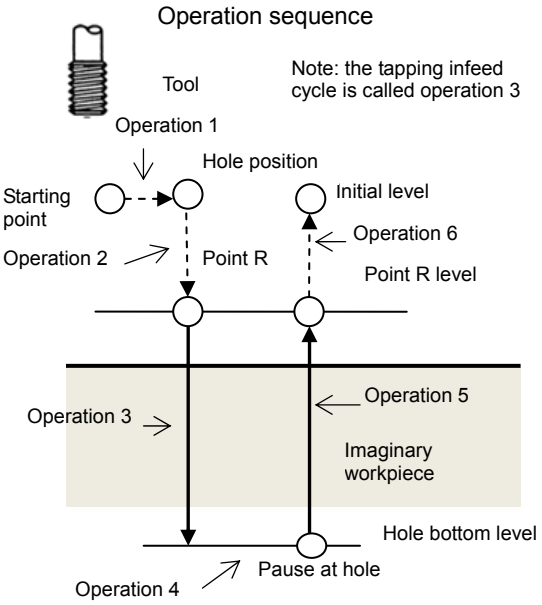


Fig. 2-76

Note 7: The tapping cycle temporarily cancels the tool nose radius compensation, and it recovers when the fixed cycle is cancelled.

Note 8: When the fixed cycle is cancelled in rigid tapping, the used S value is also cleared (its state is the same that of the specified S0). The specified S for rigid tapping cannot be used in the program followed by the one for cancelling rigid tapping .

Note 9: Specify S again after cancelling the fixed cycle of rigid tapping.

Note 10: N0.5209#0=0, i.e. drilling axis is selected by the levels in rigid tapping mode". When G17, G18, G19 is separately specified in G84, the drilling axes separately correspond to the basic axis X, Z, Y; when G17, G18, G19 is separately specified in G88, the drilling axes separately correspond to the basic axis Y, X, Z.

Note 11: In default condition, when G84/88 executes the rigid tapping infeed, the spindle rotates forward; when the tool retraction is executed, the spindle does CCW. In some special applications, the spindle rotates reversely, but the spindle does forward when the tool retracts. When the reverse thread tapping is needed, GSK988T uses the selection signal of the rigid tapping spindle rotation (RGROD, i.e. G61.2 of PLC address) to realize the reverse thread tapping.

Before G84/G88 is executed, the CNC checks the selection signal's state of rigid tapping spindle rotation to confirm the rotation of tapping axis. When RGROD signal is set to 1 and G84/G88 executes the infeed, the spindle rotates forward; when the tool retracts, the spindle rotates reversely, which is the normal thread tapping; when RGROD signal is set to 1 and the infeed is done, the spindle rotates reversely, but when the tool retracts, the spindle rotates forward, which is the reverse thread tapping. After the CNC is turned on, RGROD signal is default to 0.

In the course of G84/G88 rigid tapping, RGROD state is not changed. After G80 is executed, RGROD state can be reset. Or reset it before G84/G88 is executed.

Adding RGROD signal to the PLC ladder can realize the reverse thread rigid tapping.

Program example:

Suppose that the current system is ISB, and its least input unit is 0.001 mm.

```
G98                          ; feed per minute
M29 S1000                    ; switch to the rigid tapping mode, command the
                               spindle speed 1000. After the block is executed,
                               the spindle does not rotate.
G0 X50 Z0                    ; X and Z position to the stating point
G84 Z-50 P3000 F2000         ; starting point is X50 Z0，and the hole position is
                               the same that the starting point,
                             ; the hole bottom position is X50 Z-50，and the
                               pause time is 3s，
                             ; the thread lead is 2 according to the commanded
                               F and S value.
                             ; Q is not commanded and is the standard rigid
                               tapping cycle.
G80                          ; the fixed cycle is cancelled and the motive tool
                               stops rotation.
```

M28 ; suppose M28 is for the spindle cancelling rigid tapping mode

M30 ; End of program

2.20.3 End Common Tapping Cycle (G84) /Side Common Tapping Cycle (G88)

When G84/G88 executes the common tapping, the miscellaneous function controls the spindle start/stop: M03(spindle CW), M04(spindle CCW) and M05 (spindle stop); the CNC checks the spindle rotation based to the spindle encode and the tapping axis rotates along with the spindle. When the machine cannot use the rigid tapping function, the common tapping mode provides an economical tapping method.

The spindle must use the flexible chuck or the tool uses the variable screw tap in the common tapping mode.

Command function: when the spindle rotates one rotation, Z axis moves one pitch, which keeps consistent with the pitch of screw tap and forms one helical grooving in inner of the workpiece to complete the thread machining of inner hole one time. Pay more attention to the difference between it and the spindle rigid tapping.

Command format：G84 X (U)_ C (H)_ Z (W)_ R_ P_ F_ K_ M_ ; or
G88 Z (W)_ C (H)_ X (U)_ R_ P_ F_ K_ M_ ;

Command explanation：

X_ C_ or Z_ C_	It is the hole position data and is valid only in the specified block; the hole position data can specify other valid axes except for X, Z, C.
Z(W)_ or X(U)_	It specifies the coordinate value of hole bottom by using absolute coordinate, or specifies the distance from R level to the hole bottom by using incremental value, and it is valid in the specified block.
R_	It is the distance from the initial level to point R and is specified by radius value with direction. Its unit and range is shown below.
P_	Hole bottom pause time. Unit of ISB system is 1ms and ISC is 0.1ms.
F_	Cutting feedrate，
K_	Execution times of program（it is used when it is needed）.
M_	M command for clamping C axis（it is used when it is needed）.

+	Incremental system	Metric input（mm）	inch input (inch)
R	ISB system	-99999.999～99999.999mm	-9999.9999～9999.9999 inch
	ISC system	-9999.9999～9999.9999 mm	-999.99999～999.99999 inch

Tapping feed axis specifies X or Z axis according to G84/G88. G84 specifies Z to be the tapping axis and G88 specifies X. The spindle is selected according to relevant G signals (it is related to PLC programs).

Cutting feedrate F (i.e. feedrate of tapping axis) and spindle speed S confirm the thread

lead.

Thread lead formula in per minute mode =cutting feedrate F/spindle speed S;
Thread lead formula in per rotation mode=cutting feedrate F.

Note: The spindle speed S is defined to be S modal value memorized by the CNC before the common tapping,

The thread lead is counted by the specified F value. The spindle override is affected by N0.3708#6 in common tapping.

In G84/G88 rigid tapping cycle, Q command and PCP determine three kind of rigid tapping mode: standard rigid tapping cycle, high speed deep hole rigid tapping cycle and deep hole rigid tapping cycle. There is only one mode in G84/G88 common tapping cycle, which is shown below.

The spindle rotating(the operator confirms the CW/CCW according to the used screw tap) is specified before G84/G88 is commanded, and the CNC confirms the M command of spindle CCW according to the previous spindle rotation direction of G84/G88; when the direction is not specified, the spindle rotation (M03 CW) is defaulted in G84/G88 common tapping cycle.

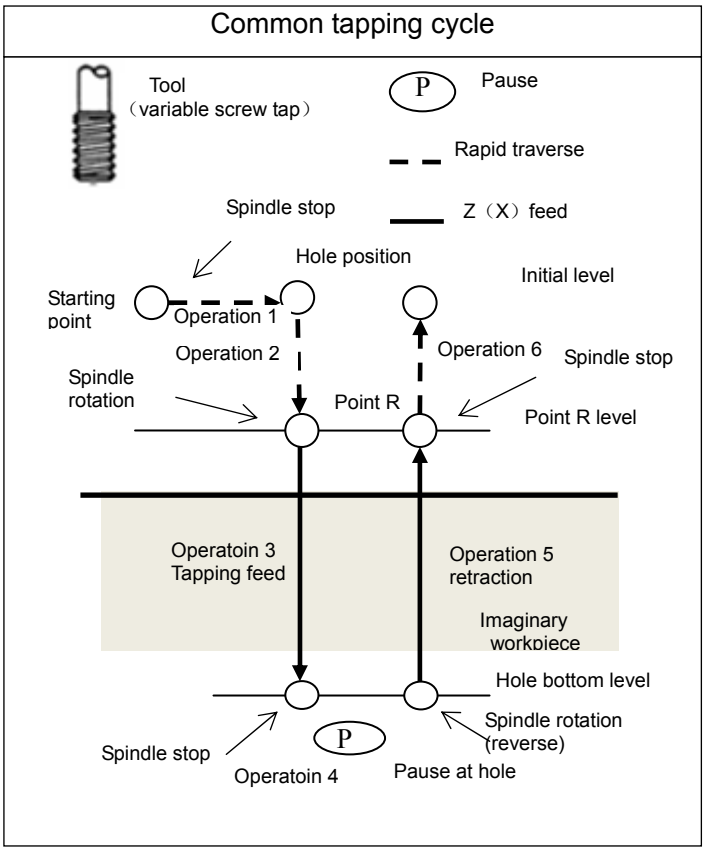


Fig.2-77

Execution process:

①The tool rapidly positions to the hole from starting point（the hole is determined by the hole position data at the initial level）;

②Rapidly position to point R;

③M command outputs to make the spindle rotate (for example, M command does not output when the spindle rotation is commanded before tapping cycle) ;

④The tapping axis moves the hole bottom level at the cutting feedrate specified by F along with the spindle rotation;

（when the tool is about to reach the hole bottom position specified by the programmed, the spindle stops M05 output and the spindle starts to decelerate, the tapping axis holds feed until the spindle exactly stops rotation）

⑤Pause is executed in the time specified by P;

⑥The spindle's rotation M command (CW) outputs (the spindle rotation direction is reverse to the tool infeed);

⑦The tapping axis returns to point R level at the speed specified by F;

⑧The spindle stops M05 output and rotation;

⑨Return rapidly to the initial level;

⑩Common tapping cycle ends.

Note 1：The command is for the flexible tapping and the tapping axis rotates along with the spindle rotation. After the spindle stop signal M05 at the hole bottom is valid, the spindle stops rotation in some deceleration time, at the moment, Z feeds along with the spindle rotation until the spindle exactly stops. The hole bottom position is deeper or lower than the actual programmed position, and the concrete error length is determined by the spindle speed and spindle brake device in the course of tapping.

So, before tapping in G84/G88, the operator moves the slider to the safety position, and the system executes G84/G88 without cutting the workpiece (it is not dry run). The operator actually observes the coordinate difference between the position where the spindle stops at the hole bottom and G84/G88 starting point, and then modifies the program to reserve the enough hole depth before G84/G88 is executed.

Note 2：Before the tapping cycle is executed, the spindle's rotation direction (i.e., command the spindle rotation(CW or CCW) before the common tapping) can be specified according to the screw tap's rotation direction, the system starts to tap after the tool reaches point R, at the moment, the CNC does not output spindle rotation M command, automatically counts the spindle rotation M command in CCW direction after the tool reach the hole bottom. G84/G88 is executed in the next block, and the tool has reached point R, the CNC again outputs the spindle rotation M command, at the moment, the spindle's rotation direction is consistent with that of the previous specified.

The CNC defaults to be spindle rotation (CW M03) when the spindle rotation is not specified. After the fixed cycle is cancelled, the spindle stops rotation. Restart the spindle when the machining is needed continuously.

Note 3: The traverse speed of tapping axis is determined by the spindle speed and pitch instead of the cutting feedrate override; the spindle override is affected by N0.3708#6.

Note 4：When the single block runs or the system executes the feed hold, the system displays "Pause" but the tapping cycle does not stop until the tapping is completed and the tool returns to the starting

point.

Note 5：The tapping cut decelerates to stop when the system resets, emergently stops or the drive unit alarms. The spindle is in the course of stopping rotation but Z has stopped feed, which maybe damage the workpiece and screw tap. So, do not force to stop G84/G88.

Note 6：N0.5209#0=0, i.e. drilling axis is selected by the levels in rigid tapping mode". When G17, G18, G19 is separately specified in G84, the drilling axes separately correspond to the basic axis X, Z, Y; when G17, G18, G19 is separately specified in G88, the drilling axes separately correspond to the basic axis Y, X, Z.

Program example: machining thread M10×2 is shown below:

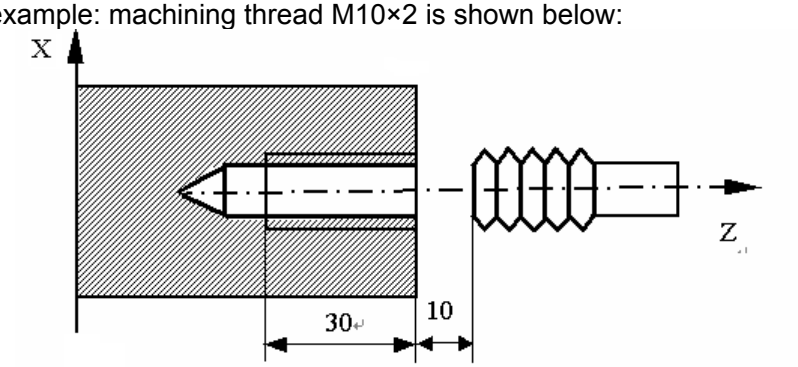


Fig. 2-78

```
G98                    ；feed per minute
G0 X0 Z200             ；X and Z position to the starting point
M3 S800                ；the spindle rotates (CW) at the speed 800 r/min. the spindle
                         starts rotation after the block is executed.
G84 Z160 P1000 F1600   ；starting point is X0 Z200，which is same as the hole position，
                       ；hole bottom position is X0 Z160, and the pause time is 1s，
                       ；the thread lead is 2 according to F and S value.
                       ；G84 is the common tapping cycle when it is not specified in
                         advance.
                       ；the spindle stops rotation after the block execution ends.
G80                    ；Fixed cycle is cancelled
M30                    ；End of program
```

2.21 Automatic Chamfering Function

Command function: Automatic chamfering function is defined to automatically insert chamfering block or coring R block between machining blocks.

Blocks where the automatic chamfering can be inserted:

- Between linear interpolation and linear interpolation
- Between linear interpolation and arc interpolation

- Between arc interpolation and linear interpolation
- Between arc interpolation and arc interpolation

Command format： ，C_ ；（chamfering）

，R_ ；（corning R）

Command explanation: one chamfering block or corning R block is inserted when the above format is specified at the end of the specified linear interpolation (G01) or arc interpolation (G02, G02) block.

Note: The system can continuously specify more than two chamfering blocks and corning R blocks.

+	Incremental system	Metric input（mm）	Inch input (inch)
,C	ISB system	-99999.999～99999.999 mm	-99999.999～99999.999 inch
	ISC system	-9999.9999～9999.9999 mm	-9999.9999～9999.9999 inch
,R	ISB system	-99999.999～99999.999 mm	-99999.999～9999.9999 inch
	ISC system	-9999.9999～9999.9999 mm	-9999.9999～999.99999 inch

Chamfering：The numerical value following C specifies the distance from chamfering starting point to end point of the imaginary cornering intersection which is defined to the imaginary existing cornering when the chamfering is not executed.

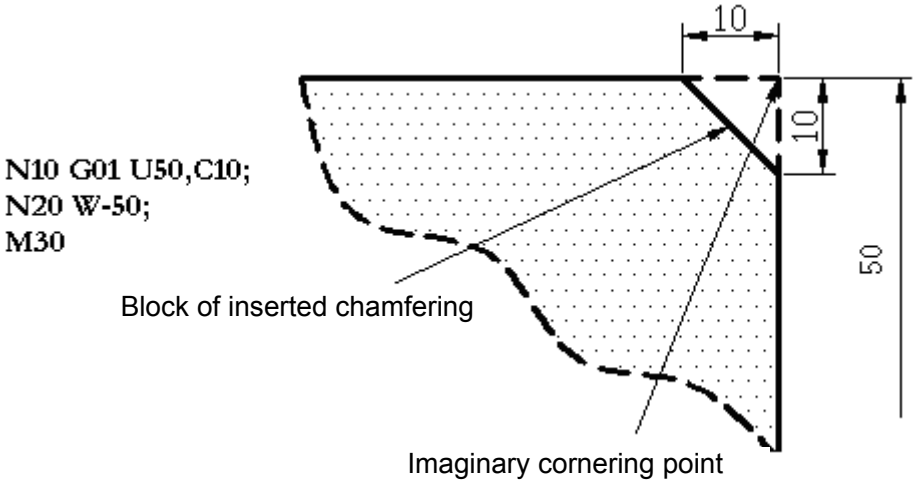


Fig. 2-79

Corning R: The numerical value following R specifies corning R radius.

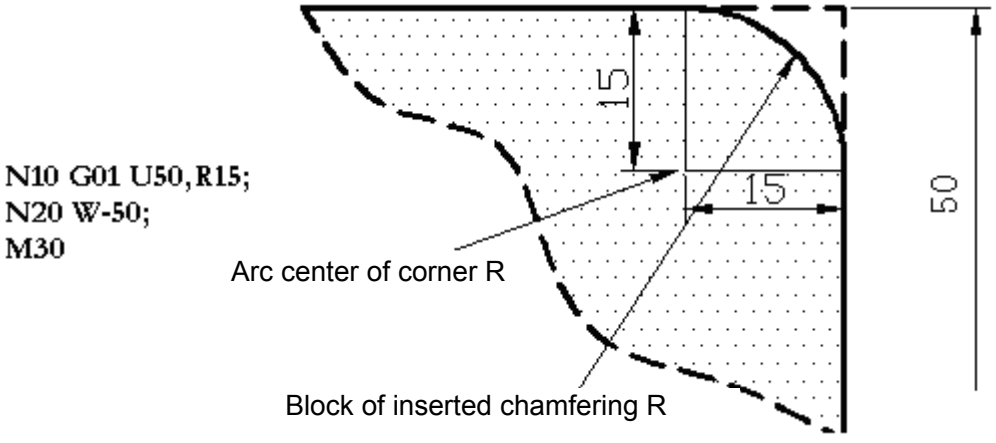


Fig. 2-80

Note 1: Even if the chamfering (, C) or corning R(,R) is specified in other blocks besides G01 and G02/G03 (except for G32, G34), it is ignored.

Note 2: The block following chamfering or corning R for the chamfering or corning operation must be the one of G01 or G02/G03. The alarm "no movement after chamfering/corning R" occurs when other commands are specified.

But, only one G04 (dwell) block can be inserted between these blocks. The system pauses after the inserted chamfering/corning R block is executed.

Note 3: When the system exceeds the previous interpolation movement range caused by the inserted chamfering or corning R block, the alarm "commanded movement being excessive small in the block following chamfering/corning R" occurs.

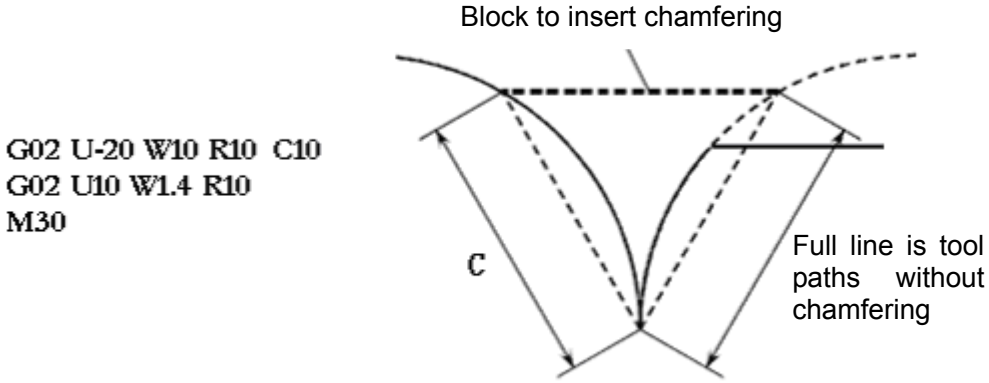


Fig. 2-81

Note 4: The chamfering or corning R block can be inserted into the movement commands included in the same level.

When the level selection (G17, G18, G19) in the next block after the chamfering or corning R is specified, the alarm occurs "the level selection command is specified after chamfering or corning R".

Note 5: When two linear interpolation operations are executed and their angle difference is within ±1, the movement of chamfering/coring R block is 0. When linear interpolation and circular interpolation operations are executed and angle difference of their tangent at the intersection point is within ±1, the movement of corning R block is 0. When two circular interpolation operations are executed and the angle difference of their circular tangent is within ±1, the movement of corning R block is 0.

Note 6: When the chamfering or corning R block is specified in a single block, the operation runs until it reaches the end point of new chamfering/corning R block, the machine stops in feed hold mode at the end point.

Note 7: The following G commands cannot be used with the chamfering/corning R command in the same block, as well as the blocks of chamfering/corning R of the defined continuous graph.
G commands in Group 00 (except for G04)

Note 8: When ",C" or ",R" is commanded in the thread cutting block, the alarm occurs "cannot command the chamfering or corning R in the current block".
Note 9: The last is valid when the many ",C" and ",R" are specified in the same block.

2.22 Macro Command

GSK988T provides the macro command which is similar to the high language, and can realize the variable assignment, and subtract operation, logic decision and conditional jump by user macro command, contributed to compiling part program for special workpiece, reduce the fussy counting and simplify the user program.

2.22.1 Variable

（1）variable use

The variable can specify the address value in the program. The variable value is assigned by the program command or is set directly by the keyboard. One program can use many variables which can be distinguished by their variable number.

- **Variable expression**

  Use "#"+variable number to express;

  Format：**# i**（i=200，202，203，……）；

  Example：#205，#209，#225.

Besides, the expression can be used to specify the variable number. At the moment, the expression must be in the brackets.

Example：#[#20+#30/4]

- **Variable reference**

1. Use variable to permute the number following address

Format: ＜address＞+"# i" or ＜address＞+"－# i" means to take the variable value or the negative value of value of the variable as the address value

Example: F#203…#203=15: it is the same those of F15 functions;

Z-#210…#210=250: it is the same those of Z-250 functions;

G#230…#230=3: it is the same those of G3 functions.

When the variable value is used in program, the decimal point can be omitted. Example: #1=123: the actual value of #1 is 123.000.

When the variable value followed the axis command address has the decimal point, the data less than the least setting unit executes the rounding. For example: #1=1.23456; the axis least setting unit is 0.001, the tool to execute G00 X#1 positions to 1.235 position.

2. Use variable to permute variable number.

Format: "#"+[variable number]

Example: 5 uses #30 to execute the permutation in #5, is written to #[#30].

3. Refer the undefined variable.

When the variable is not defined, it becomes the "Null" variable. When the variable #0 is Null, it is only read instead of being written.

When the system refers to the undefined variable, it ignores the variable and the word.

Example: when the variable #10 value is 0, the variable #!1 value is Null and the system executes G00 X#10 Y#11, the execution result is G00 X0，Y#11 to be ignored.

Beside using the Null to assign, the variable value is 0 in other conditions.

When #2=< Null >时，#1=#2, #1=<Null>；

#1=#2·3, #1=0;

#1=#2+#2, #1=0;

<Null> in conditional expression is different with 0.

When #2=<Null>, #2 EQ #0，#2 NE 0, the condition is tenable.

When #2=0, #2 EQ #0，#2 NE 0, the condition is not tenable.

（2）Variable Type

The variable is divided into the different variable types according to the variable number, their use and prosperity are different as follows:

Variable range	Variable type	Function
#0	Null variable	The variable is null and is not assigned.
#1～#33	Local variable	The local variable is used to store data in the macro program, such as result. When the system is turned off, the local variable is initialized to be null. When the macro program is called, the argument assigns to the local.
#100～#199 #500～#999	Share variable	The share variable has the same meaning in the different macro program. When the system is turned off, the variable #100~#199 is initialized to be null, #500~#999 is saved and is not lost.
Behind #1000	System variable	The system variable is used to read all types of data when CNC runs.

（3）Variable range

The input range of the local variable and common variable is -99999999~99999999 which integer part and decimal part are up to 8-digit number. The system alarms when the assignment exceeds the valid range. The system alarms when the assignment value exceeds its range. The middle result in the macro variable count can be more than the valid input digital.

Note 1: The variable cannot be referred to address O and N. The system cannot use O#200，N#220 to execute the programming;

Note 2: When the variable exceeds the max. command value defined by the address, it cannot be used; for example: #230 = 120: M#230 exceeds the max. command value;

Note 3: The system cannot identify –0 and + 0. # 4 = - 0: X # 4 is taken as X 0;

Note 4: When the variable is used to the address data, the other except for the valid digit is rounded.

Note 5: The number followed by the address can use <Formular> to replace. The system takes "Word address [<Formular>]" or word address-[<Formular>]" as a program, and take <Formular> value or its negative value as the command value of the address.

Note 6: The decimal point which defines the variable in a program can be omitted. For example, #1=123 is defined, the actual value of #1 is 123.000;

Note 7: The negative sign of variable value which changes the reference should be placed in the front of #, such as G00X-#1;

Note 8:The variable #1～#33, #100～#199 are cleared out after they reset, which are set by NO.6001Bit7（CLV） and Bit6（CCV）, and which cannot be executed in MDI mode;

Note 9: When the variable value overflows, the command address referring to the variable is ignored.

Note 10: NO.6000 Bit5（SBM）sets whether the single block stop is valid in user macro program.

Note 11: The number in expression (including brackets) can be omitted. For example, X[10] actual value is X10.000.

2.22.2 System variable

The system variable is used to read and write NC internal data. For example, some system variable only read the tool offset value and current position data. The system variable is the base of the automatic control and general machining program development.

（1）Interface signal

The interface signal can program the exchange message between the machine controller and user macro programs, i.e. it completes the exchange with PLC by G, F signals and the interfaces

with IO are defined by PLC.

The input signal can be only read, and the output signal can be read and written.

Variable number	System variable of interface signal	
	Function	Corresponding G, F signals
#1000 ～ #1015	Read the signal with 16 bits according to its bit from PLC to user macro program.	corresponding to G54.0～G54.7, G55.0～G55.7 signal states
#1032	Read the signal with 6 bits one time.	Corresponding to G54,G55 signal states
#1100 ～ #1115	Write the signal with 16 bits according to its bit to PLC.	Corresponding to F54.0～F54.7, F55.0～F55.7 signal states
#1132	Write the signal with 16 bits to PLC one time.	Corresponding to F54, F55 signal states
#1133	Write the signal with 32 bits to PLC one time. Specify from -99999999 to +99999999	Corresponding to F56, F57, F58, F59 signal states

（2）Tool compensation value

The system variable can read/write the tool compensation value. The system variable of the tool compensation storage area is 1501～2999. The variable numbers divided exactly in the above range are illegal. The variable number of 2201～2299，2901～2999 alarm. The concrete range are referred to the following table.

Set the axis number to be n(1～5), the compensation number to be m(1～99), the offset variable number of the axis to be 1600+(n-1)*100+m, the wear variable number to be 2300+(n-1)*100+m.

Compensation number	1st axis		2nd axis		3rd axis		4th axis	
	Offset	Wear	Offset	Wear	Offset	Wear	Offset	Wear
1	1601	2301	1701	2401	1801	2501	1901	2601
…	…	…	…	…	…	…	…	…
99	1699	2399	1799	2499	1899	2599	1999	2699

Compensation number	5th axis		Radius compensation value R		Tool nose T
	Offset	Wear	Wear	wear	
1	2001	2701	2101	2801	1501
…	…	…	…		…
99	2099	2799	2199	2899	1599

Note: Range of #1501-#1599: 0-9, and is rounded when it is with decimal point.

（3）Marco program alarm

There is the alarm and the alarm message specified by the user in program. The variable is only written instead of being read.

Variable	Function
#3000	When the system executes the assignment statement of #3000=XXX, it stops the run and alarms.

	The alarm message only displays 26 characters (13 Chinese characters), and the system only displays the first 26 characters when there are more than it. The value of the alarm number being #3000 adds 3000, the alarm range is 3000 to 3200. When #3000 value is less than 0, the alarm number is 3000, when #3000 value is more than 200, the alarm number is 3200.

Example:

#3000=6; the tool has not found

When the system executes the block, it stops and alarms and the alarm number is 3006. The alarm message is “TOOL NOT FOUND”, The system maybe alarm in advance because of the buffer exists.

The alarm message can use the small brackets. For example, #3000=6(TOOL NOT FOUND). When the small brackets and the semicolon are in the block, the latter specified message is valid, such as #3000=6(TOOL NOT FOUND); TOOL NOT FOUND，the displayed message is “TOOL NOT FOUND”.

（4）Stop message

The program execution is interrupted and the system displays one message. i.e. the single stops after the system executes the block, and the system displays only one prompt. The variable is only be written instead of being read.

Variable	Function
#3006	When the system executes the assignment statement of #3000=1, it stops the run and displays only one prompt message. The alarm message only displays 26 characters(13 Chinese characters), and the system only displays the first 26 characters when there are more than it. The value of the alarm number being #3006 adds 3200, the prompt number range is 3201 to 3500. When #3006 value is less than 1, the alarm number is 3201, when #3006 value is more than 300, the alarm number is 3500.

For example:

#3006=3; wait for run

When the system executes the block, it stops and displays one prompt and the prompt number is 3206. The prompt message is “WAITING FOR RUN”. The format of the prompt message is the same that of description in the macro program alarm.

（5）Machine workpiece quantity

The required workpiece quantity and machined workpiece quantity are read and written

required workpiece quantity and machined workpiece quantity	
Variable	Function
#3901	Machined workpiece quantity(completed quantity)
#3902	Required workpiece quantity(target quantity)

When #3901 value is changed, the workpiece quantity displayed in POSITION window also changes.

When #3902 value is changed, No.6713 value also changes.

（6）Modal message

The previous modal message which is being processed can be read.

Variable number	Function	
#4001	G00, G01, G02, G03, G32, G34, G90, G92, G94	No. 1 group
#4002	G96, G97	No. 2 group
#4003		No. 3 group
#4004		No.4group
#4005	G98, G99	No.5group
#4006	G20, G21	No.6group
#4007	G40, G41, G42	No.7group
#4008	G25, G26	No.8group
#4009	G22, G23	No.9group
#4010	G80, G84, G88	No.10group
#4011		No.11group
#4012	G66, G67	No.12group
#4013		No.13group
#4014	G54, G55, G56, G57, G58, G59	No.14group
#4015		No.15group
#4016	G17, G18, G19	No.16group
◦◦◦		◦◦◦
#4022		No.22group
#4109	F command	
#4113	M command	
#4119	S command	
#4120	T command	

Example:

When the system executes #1=#4016, #1 value is 17, 18 or 19.

The system alarms when the reading/writing modal value is G command which cannot be used by the system.

（7）Current position

The position message is only read instead of being written.

Variable number	Position signal	Coordinate system	Tool compensation value
#5001--#5005	End point of block(absolute coordinate)	Workpiece coordinate system	Not including
#5021--#5025	Current position(machine coordinate)	Machine coordinate system	including
#5041--#5045	Current position(machine coordinate)	Workpiece coordinate	including

		system	
#5061--#5065	Skip signal position	Workpiece coordinate system	including
#5081--#5085	Tool length compensation value		

The read is the position value after the last block execution.

The units digit from 1 to 5 of variable number corresponds the No. n axis.

（8）Compensation value of workpiece coordinate system

The workpiece zero offset value can be read and written.

Variable number	Function
#5201--#5205	External zero offset value
#5221--#5225	G54 workpiece zero offset value
#5241--#5245	G55 workpiece zero offset value
#5261--#5265	G56 workpiece zero offset value
#5281--#5285	G57 workpiece zero offset value
#5301--#5305	G58 workpiece zero offset value
#5321--#5325	G59 workpiece zero offset value

The units digit from 1 to 5 of variable number corresponds the No. n axis.

（9）Note

The system variable is the state value of the system, and is buffered in advance when multi cycles are executed, so, the attained system variable is the value before the multi cycle command instead of the current value to avoid using the system variable in the cycle body of the multi cycles.

2.22.3 Operation and jump command

（1）Operation command

Variables can execute all kinds of operations, and their operation command format is as following.

#i=＜Expression＞

The right <expression> of an operation command is a compose of constant, a variable, function and operator.

GSK988T defines the following operations and logic commands:

Function	Format	Use
assignment	#i=#j;	Assignment statement assigns #j value to #i； #i is Null when #j is Null;
addition	#i=#j+#k;	Addition. When #j value is Null, it it taken as 0.0 value, and the following functions are the same that of it;
Subtraction	#i=#j-#k;	Execute subtraction operation;
Multiplication	#i=#j*#k;	Execute division operation;
Division	#i=#j/#k;	Execute addition;
Sine	#i=SIN[#j];	Execute sine operation; Angle unit is degree;

Arc sine	#i=ASIN[#j];	Execute arc sine operation; #j value is from -1 to 1
cosine	#i=COS[#j];	Execute cosine operation ; Angle unit is degree;
Arc cosine	#i=ACOS[#j];	Execute arc cosine; #j value is from -1 to 1 Function range: 0°～180°
Tangent	#i=TAN[#j];	Execute tangent operation ; Angle unit is degree; #j value cannot be 0，90，270
Arc tangent	#i=ATAN[#j]/[#k];	Specify the lengths of two sides, execute the arc tangent, #j is opposite with “/” to partition;
Square root	#i=SQRT[#j];	Execute square root operation; #j cannot be less than zero
Absolute value	#i=ABS[#j];	Execute absolute value operation;
Rounding	#i=ROUND[#j];	Execute rounding operation; In macro program, execute the rounding of one-digit of No., in NC statement, execute the rounding of the next digit of the least increment
FUP	#i= FUP [#j];	Floating UP integer In puls quantity, #i is more than or equal to #j，in the negative,#i is less than or equal to #j
FIX	#i= FIX [#j];	Floating FIX integer In puls quantity, #i is less than or equal to #j，in the negative,#i is more than or equal to #j
Natural logarithm	#i=LN[#j];	Execute natural logarithm The system alarms when #j is zero or less than zero
Exponential function	#i=EXP[#j];	Execute #j exponent #j value cannot be more than 80;
OR	#i=#j OR #k;	Execute the binary logic operation of input data #j, #k cannot be less than zero When there are the decimal points in #j, #k, the decimal parts are rounded
XOR	#i=#j XOR #k;	
AND	#i=#j AND #k;	
BCD to BIN	#i=BIN[#j];	Converse the decimal data into the binary The system alarms for the data which cannot the converse
BIN to BCD	#i=BCD[#j];	Converse the binary into the decimal

Command explanation:

（1）operation sequence:

Prior	Operator and function
5	"" [" , "]"
4	"#"
3	"SIN", "SI", "ASIN", "AS", "COS", "CO", "ACOS", "AC", "TAN", "TA", "ATAN", "AT", "SQRT", "SQ", "ABS", "AB", "ROUND", "RO", "FIX", "FI", "FUP", "FU", "LN", "EXP",

	"EX", "BIN", "BI", "BCD", "BC",
2	"AND", "AN", "*", "/",
1	"OR", "XOR", "XO","+", "-",

（2）EXP function input value cannot be more than 80, otherwise, the system alarm;

（3）"/" character in <expression>(in the right of assignment"=" or in the bracket []) is taken as the division operator instead of optional block skip code;

（4）The bracket "[]" can use 5-level, including the used bracket in the function, and the system alarms when it exceeds 5-level;

（5）The angle units of the triangle function SIN, COS, ASIN, ACOS, TAN and ATAN are degrees, for example: 90°30′ is 90.5 degree;

（6）#i=ASIN[#j] value range:

When NO.6004 No. 0-digit NAT is set to 0: 90°～270°

When NO.6004 No. 0-digit NAT is set to 1: -90°～90°

When #j exceeds between -1 and 1, the system alarms and #j can be a constant.

（7）#i=ACOS[#j] range: 0°～180°。

When #j exceeds between -1 and 1, the system alarms and #j can be a constant.

（8）In #i= ATAN[#j]/[#k], ATAN #j and #k are the weight length of two right-angle sides as follows:

When NO.6004 No. 0-digit NAT is set to 0: 0°～360°

Example: when #1=ATAN[-1]/[-1] is specified, #1=225°.

When NO.6004 No. 0-digit NAT is set to 1: -180°～180°

Example: when #1=ATAN[-1]/[-1] is specified,#1=-135°

#j，#K can be the constant.

In division or TAN[90], the division is specified to 0, P/S alarms;

（9）The function ROUND is used to NC command or macro statement, which rounds the data with the decimal point. It is used to NC statement, which rounds according to the least setting unit; when it is used to the macro statement, which rounds No. 1-digit decimal point;

In executing #2=ROUND[#3], when #3=1.2345, the variable #2 value is 1.

In ISB increment metric input, #2=1.2345, #3=2.5456:

G00 X#2； the tool moves to 1.235mm

G00 X#3； the tool moves to 2.546mm

（10）For FUP, FIX, when the absolute value of the integer after execution is more than that of the original, it is FUP; when it is less than that, it is FIX.

When #2=1.2，#3=-1.2

In executing #4=FUP[#2], 2.0 is assigned to #4

In executing #4=FIX[#2], 1.0 is assigned to #4

In executing #4=FUP[#3], -2.0 is assigned to #4

In executing #4=FUP[#3], -1.0 is assigned to #4

（11）Logic operation OR, XOR, AND firstly are conversed the decimal into the binary, and are executed in the binary by one-digit to one digit.

Range: 0～99999999，when it has the decimal point, it is ignored.

Example:

#101=10 （the binary is: 00001010）

#102=12 （the binary is: 00001100）

#103=#101 OR #102 （or the operation result is : 00001110）

The window display result of macro variable is #101=10.000000 #102=12.000000 #

103=14.000000

（12）The function BIN converses the decimal into the binary displayed in 8421 format BCD. The system cannot display and alarms when some digit in BCD code after conversion exceeds 9.

The function BCD converses the BCD code displayed in 8421 format into the decimal.

Example 1：

#101=55 （The binary: 00110111）

#102=BIN[#101]

Macro variable window display #102=37.000000

Example 2：

#101=37 （BCD 37 corresponds to the binary : 00110111）

#102=BCD[#101]

Macro variable window display #102=55.000000

（2）Transfer and repetition commands

The transfer and the repetition commands can change the control flow, and there are three kind of transfer and repetition operation: the unconditional transfer GOTO, the conditional transfer IF…GOTO, IF…THEN and WHILE DO repetition.

Command format:

GOTO n;

Command function:

Skip to the line number n without condition;

Command format:

IF <Logical expression> THEN <expression>;

Command function:

When the logical expression is valid, the system executes one following THEN, otherwise, it executes the next block.

Command format:

IF < Logical expression > GOTOn;

Command function:

When the logical expression is valid, the system skips the block with the line number n to execute, otherwise, it executes the next block;

Command format:

WHILE < Logical expression > DOn;

……;

ENDn

Command function:

When the logical expression is valid, the system executes the block between Do and END, otherwise, its execute the block following END. The numerical value n following DO and END is used to specify the execute range label of the specified program, n value is 1, 2, 3. The system alarms when n is not 1, 2, 3.

IF, WHILE logical operation character rules are as follows:

Operator	substitute	character definition
EQ	==	（=）
NE	<>	（≠）
GT	>	（＞）
GE	>=	（≥）
LT	<	（＜）
LE	<=	（≤）

Note 1: When the system transfers to the block with the serial number n and specifies the another exceeding the serial number range between 1 and 99999, P/S alarms, and the expression can specifies the serial number;

Note 2: The conditional expression must include the operator which is inserted in the middle of two variables or the variable and the constant and is closed by the bracket[]. The expression can be replaced by the variable;

Note 3: The number following D0 and the one following END specify the execution range label of the specified program, and the label value is 1, 2, 3. The system alarms when n is not 1, 2, 3;

Note 4: The label (1-3) in the repetition DO—END can be used many times, but P/S alarms when there is the cross repetition(superposition in DO range);

Note 5: When the system specifies D0 instead of WHILE statement, it creates the limitless repetition between DO and END;

Note 6: In using EQ, NE logical operation expression, <Null> and zero have the different result. <Null> is taken as the zero in +, -, * conditional expression;

Note 7: The macro program statement cannot be used with NC statement together, and the macro program statement definition is as follows:

Block including arithmetic or logical operation(=);
Block including the control statement(such as TOTO, DO, END);
Block including macro program call command(such as G65, G66, G67 or other G codes, M code call macro program) ;
Any blocks except for macro program statements are NC statements;

Note 8: Any blocks except for macro program states are NC statements.

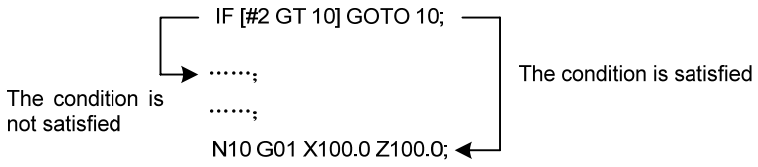
Note 9: The system can use the substitution character which is easily understood to replace the operator. '>', '<' can be edit in PC instead of on MDI keyboard and are uploaded into the system;

Note 10: When macro statement needs a line number, the line number must be compiled in the front the statement;

Note 11: In MDI mode, the system cannot execute the skip statement, otherwise, it alarms.

Example:

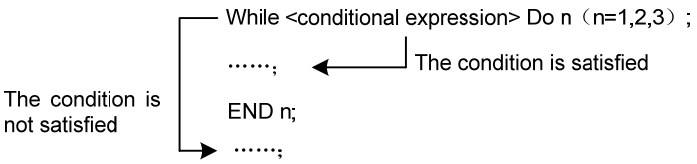
（1）GOTO example



（2）IF <Logical expression> THEN <expression> example

IF[#2 EQ #3] THEN #4=0；
When #2 value is same that of #3, #4 value is 0.

（3）WHILE <Logical expression> DOn;…；ENDn example



2.22.4 Macro program statement and NC statement

The following blocks are macro program statements:

Including arithmetic or logical operation (=);

Including control statement(such as GOTO, DO, END);

Including macro program call command (G65, G66, G67).

Any NC blocks except for macro program statement are NC statements.

In Single Block mode, when No.6000 Bit5 (SBM) is set to 0, the system directly skips the macro program statement and the machine does not stop, but it is set to 1, the system stops run and enters the stop state.

One block cannot have the macro program statement and NC statement simultaneously.

2.22.5 Macro program call

（1）Non-modal call of macro program G65

Command format: G65 P__ L__ 〈argument list 〉;

Command function: The system calls macro program L times specified by P and transfers the argument to the called macro program.

Command explanations: P: specify the macro program to be called;

L：times of calling the macro program, and its default is 1 and its range is 1～9999;

Argument list: data transferred to macro programs.

Argument specification：

Two types of argument specification are available. Argument specification I uses letters other than G, L, O, N and P once each. Argument specification II uses A, B and C once each and also uses I, J, and K up to ten times. The types of argument specification is determined automatically according to the letters used.

Argument specification I

Address	Variable No.	Address	Variable No.	Address	Variable No.
A	#1	I	#4	T	#20
B	#2	J	#5	U	#21
C	#3	K	#6	V	#22
D	#7	M	#13	W	#23
E	#8	Q	#17	X	#24
F	#9	R	#18	Y	#25
H	#11	S	#19	Z	#26

Addresses G, L, N, O and P cannot be used in arguments;

Addresses that need not be specified can be omitted and local variables corresponding to an omitted address are set to null;

Addresses do not need to be specified alphabetically. They conform to word address format.

Example: B_A_D_…J_K_ Correct

B_A_D_…K_J_ Incorrect

Argument specification II uses A, B and C once each and uses I, J, and K up to ten times. Argument specification II is used to pass values such as three-dimensional coordinates as arguments.

Argument specification II

Address	Argument No.	Address	Argument No.	Address	Argument No.
A	#1	K3	#12	J7	#23
B	#2	I4	#13	K7	#24
C	#3	J4	#14	I8	#25
I1	#4	K4	#15	J8	#26
JI	#5	I5	#16	K8	#27
K1	#6	J5	#17	I9	#28
I2	#7	K5	#18	J9	#29
J2	#8	I6	#19	K9	#30
K2	#9	J6	#20	I10	#31
I3	#10	K6	#21	J10	#32
J3	#11	I7	#22	K10	#33

Note 1: G65 must be specified before any argument;

Note 2: After G65, specify at address P and L. when P or L is repeated and No.3403 Bit6 (AD2) is set 0, the specification later takes precedence, otherwise, the system alarms;

Note 3: Subscripts of I, J, K in the argument specification II for indicating the order of argument specification are not written in the actual program;

Note 4: The CNC internally identifies argument specification I and argument specification II. If a mixture of argument specification I and argument specification II is specified, the type of argument specification specified later takes precedence;

Note 5: Calls can be nested to a depth of four levels including simple calls G65 and modal calls G66. This does not include subprogram call M98.

Note 6: Whether the units used for argument without a decimal point correspond to the least input increment of each address is related to the parameter DPI (No.3401#0);

Note 7: G65, G66 cannot be in the same block with NC code, otherwise, the system alarms;

Note 8: In macro program nesting call, the local variables from level 0 to 4 are provided for nesting. When the level of the main program is 0, each time a macro is call, the local variable level is incremented by one. The values of the local variables at the previous level are saved in the CNC. When M99 is executed in a macro program, control returns to the calling program. At that time, the values of the local variables saved when the macro was called are restored.

Note 9: The line number of the command line of the macro statement must be home, otherwise, the system alarms.

Macro program nesting example

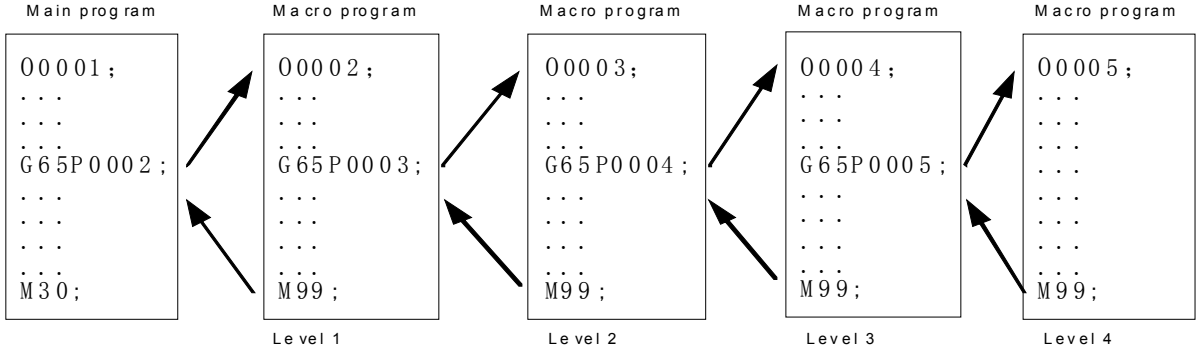


Fig.2-60 Nesting macro program

（2）Modal call of macro program G66, G67

Command format：G66 P __ L __ 〈argument list 〉；

……；

G67；

Command function: set the modal message of the specified macro program L times for calling P, send the argument to the called macro program.

Command explanation:

G66：modal macro program call needs one line to be specified;

G67：call macro program call mode;

P：specify many called macro programs;

L：times for calling the macro program. It is default to 1, its range is 1—9999;

Argument list: data sending to macro program is referred to the explanations of G65.

Note 1:Cannot call many macro programs in G66 block, but can call G66 again;

Note 2: G66 is specified before P_, L_ and argument, and the use methods of P, L, the argument are the same those of G65;

Note 3: Can't call macro program in the block without movement commands but with the auxiliary function;

Note 4: The local variable (argument) is specified only in G66 block, and the system does not set it again when each modal call is executed;

Note 5: Cannot specify the macro call command in MDI mode;

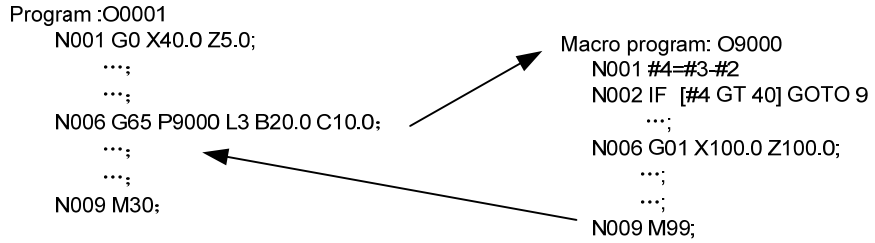
Note 6: When the reset is executed by setting the parameter, whether the common variables of the local variables from #1 to #33 and from #100 to #149 are cleared to the Null value.

Note 7: The system clears the call state of all user macro programs and subprograms and DO state, and returns to the main program;

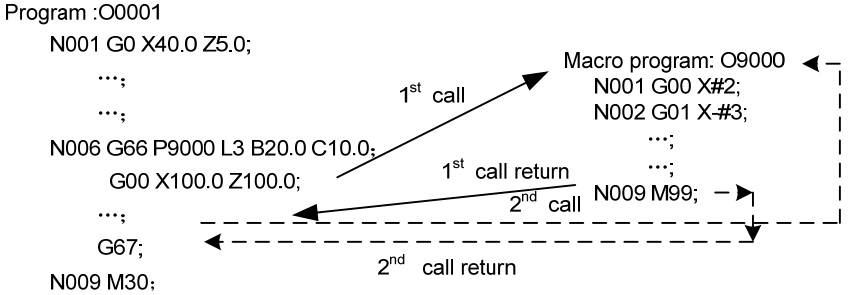
Note 8: In executing the macro program statement, when the feed pause is valid, the machine stops after the macro statement is executed, and the machine also stops when the system resets or alarms.

Application example：

（1）G65 example



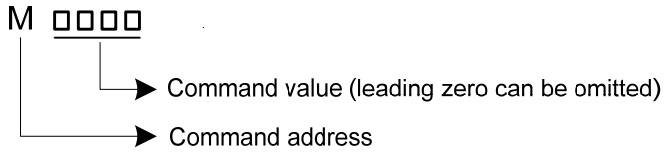
（2）G66, G67 example



Chapter Ⅲ MSTF Commands

3.1 M (Miscellaneous Function)

M command consists of command address M and its following 1～2 or 4 bit digits, used for controlling the flow of executed program or outputting M commands to PLC .



There is one valid M code in one block. There are most specified 3 M codes in one block(set by NO.3404 Bit 7 (M3B)). The corresponding relationship between M codes and their functions are determined by the machine manufacturer. CNC sends M code signal and one strobe signal to PLC in executing M codes.

Except for M98, M198, M99, all M codes are executed in PLC.

The following M codes have special meanings.

3.1.1 End of program M02

Command format: M02 or M2

Command function: In Auto mode, after other commands of current block are executed, the automatic run stops, and the cursor stops a block in M02 and does not return to the start of program. The cursor must return to the start of program when the program is executed again.

Except for the above-mentioned function executed by NC, M02 function is also defined by PLC ladder diagram as follows: current output of CNC is reserved after M02 is executed.

3.1.2 End of program run M30

Command format: M30

Command function: In Auto mode, after other commands of current block are executed in M30, the automatic run stops, the amount of workpiece is added 1, the tool nose radius compensation is cancelled and the cursor returns to the start of program (whether the cursor return to the start of program or not is defined by parameters).

Besides the above-mentioned function executed by NC, M30 function is also defined by PLC ladder diagram as follows: the system closes M03, M04 or M08 signal output and outputs M05 signal after M30 is executed.

3.1.3 Program stop M00

Command format: M00 or M0

Command function: the system stops the automatic run after M00 block is executed, which is same that of the single block pausing to save the previous modal message, i.e. which is equal to the program pause function. Press the CYCLE START key on the operation panel to execute the follow block and the CNC continuously automatically runs.

When M00 are other G command are in the same block, the system executes the command in the block, then M00, and last stops running.

3.1.4 Optional stop M01

Command format: M01 or M1

Command function: after the block containing M01 is executed, the system stops the automatic run and the single block stopping signal lights. M01 is valid when the OPTIONAL STOP on the machine operation panel is pressed.

3.1.5 Subprogram call M98

Command format：M98 P○○○○□□□□

Called subprogram number

Call times

Command function: In Auto mode, after other commands in the current block are executed in M98, CNC calls subprograms specified by P.

When the subprogram is called one time, ○○○○ can be omitted in inputting the number"○○○○□□□□" behind P, at the same time, the leading zero of the called subprogram number can be omitted and the system does not alarms. Example: M98 P12; it expresses to call the subprogram O0012 one time; the leading zero cannot be omitted when the subprogram call times are more than one.

The called subprogram name in M98 must be the program in the system and be less than 9999, and the subprogram name must be input.

The specified call times in M98 is 1～9999.

The called subprogram format in M98 is the following. The last end of the subprogram must be M99 instead of M30, its program compiling format is the same that of the main program compiling format.

Subprogram： O□□□□；（subprogram name）

…；

…；

M99；（return from subprogram）

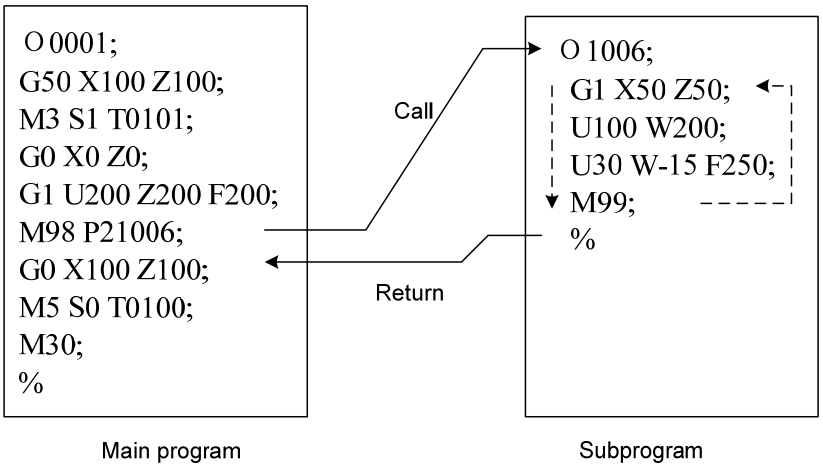


Fig.3-1 subprogram call

The called subprogram can call other subprograms. The subprogram called by the main program is called as the one-embedded subprogram, and the one called by the one-embedded subprogram is called as the two-embedded subprogram and so forth. One main program can call 12-embedded subprogram(including macro program call). The following is the four-embedded subprogram.

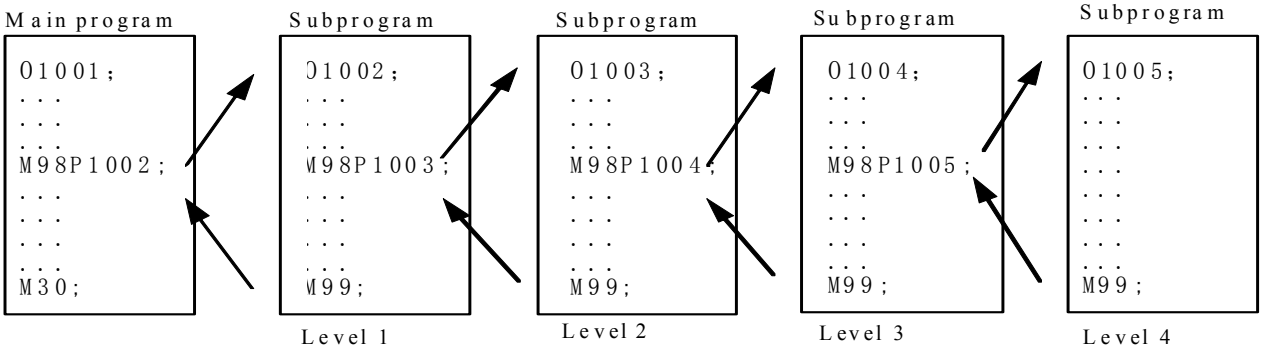


Fig. 3-2 Subprogram nesting

Note 1: The system alarms when it has not searched the subprogram specified by P;
Note 2: The system alarms when M98P__ is input in MDI, and the subprogram call cannot be executed;
Note 3: The system alarms when P98P__ call itself;
Note 4: The system alarms when M98 is commanded and the subprogram is called without P command.

3.1.6 Subprogram Call M198

Command format：M198 P○○○○□□□□

Called subprogram number in I/O

Call times

Command function: in Auto mode, when M198 is executed and the other commands in the current block has been performed, the CNC calls the subprogram in the external input/output device (usually, it is U disk) specified by P.

3.1.7 Return from Subprogram M99

Command format： M99 P○○○○○

Executed block after returning to the main program is 0000～9999，and its leading zero can be omitted.

Command function: After other commands of current block in the subprogram are executed, the system returns to the main program and continues to execute next block specified by P, and calls a block following M98 of current subprogram when P is not input. The current program is executed repeatedly when M99 is defined to end of program (namely, the current program is executed without calling other programs).

Example: Execution path of calling subprogram (with P in M99) as Fig. 3-3.Execution path of calling subprogram (without P in M99) as Fig. 3-4.

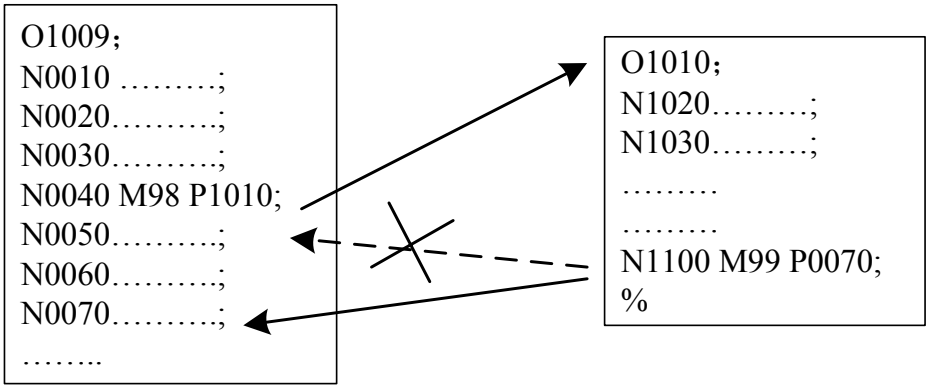


Fig. 3-3

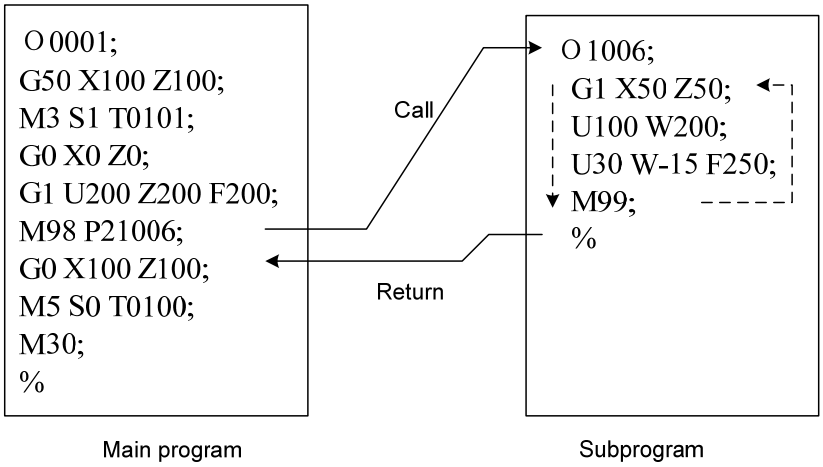


Fig. 3-4

Note 1: M99 does not need to be specified in the alone block. Example: G00 X100 Z100 M99;
Note 2: The system alarms when M99 has commanded the block number which does not exist;
Note 3: In Auto mode, the program returns to the block which is placed in the front when the specified block number behind M99 is repetitive in the program;
Note 4: In Auto mode, the system ignores the line and returns to the beginning of the file to perform the execution when the main program ends in M99 and specifies the line number following P.

3.1.8 The Following M commands for standard ladder(some functions modified by K parameters)

M3：spindle rotation (CW) M4：spindle rotation(CCW) M5：spindle stop

M8：cooling ON M9：spindle OFF

M10：tailstock forward M11：tailstock retreat

M12：chuck clamping M13：chuck releasing

M32：lubricating ON M33：lubricating OFF

M41-M44：specify gear change when the automatic gear change is performed.

M51-M58：the spindle rotates to the one of set eight positions when the spindle eight-point orientation function is valid.

3.1.9 M Commands defined by standard PLC ladder

(1) M00, M01, M02, M30, M98, M99 is separately specified in one block. When it with other M command are specified, the system ignores the other M command and the above M command is executed; when the above seven M commands are in the same block, the first commanded M command is valid.

(2) When M05, M11, M13, M33, M9 and G commands are in the same block, there are two execution methods:

a) The motion commands and M miscellaneous function commands are executed simultaneously.

b) The miscellaneous function commands following the motion commands are executed.

Refer to the tool manufacturer's user manual to selection the method.

The second method is executed for GSK's standard ladder.

(3) CNC permits there are up to specified 3 commands in one block (when NO.3404 Bit7 M3B is set to 1), some M commands cannot be specified simultaneously because of machinery operation, such as the spindle's automatic gear change commands: M41, M43, M44.

(4) No.3010 sets the delay time of the strobe signal MF, SF, TF signals.

(5) No.3011 sets the width of M, S, F function end signals (FIN).

3.2 Spindle Function

S command is used to controlling spindle speed. In GSK988T spindle speed control, NC outputs 0~10V analog voltage signal to spindle servo device or inverter to realize the gradeless spindle speed.

3.2.1 Spindle speed analog voltage control

Command format: S □□□□□

Command function: the spindle speed is defined, and the system outputs 0～10V analog voltage to control spindle servo or converter to realize the stepless timing. S command value is not reserved, and it is 0 after the system is switched on.

Command explanation: spindle speed analog voltage control command

□□□□□ means the set spindle speed, its value range is referred to Table 1-4, and the leading zero can be omitted. When the value exceeds the range set by No.3772, the most spindle speed limit is specified in the program, and S value is specified to the most spindle speed; when it is not specified, the upper and lower limit of S value is specified. The system alarms when the decimal is input to the specified of the S value. The system can set the digit number by No.3031.

The first spindle of the CNC can execute 4-gear spindle speed, and the second spindle has 2-gear spindle speed. In executing S command, the system counts the analog voltage value corresponding to the specified speed according to setting value(corresponding to No.3741～No.3744) of max. spindle speed (analog voltage is 10V)of current gear, and then outputs to spindle servo or converter to ensure that the spindle actual speed and the requirement are the same.

After the CNC is switched on, the analog output voltage is 0V. The analog output voltage is reserved (except that the system is in cutting feed in the surface speed control mode and the absolute value of X absolute coordinates is changed) after S command is executed. The analog output voltage is 0V after S0 is executed. The analog output voltage is reserved when the system resets and emergently stops.

When the spindle speed analog voltage control is valid, there are 2 methods to input the spindle speed: the spindle fixed speed is defined by S command(r/min), and is invariant without changing S command value, which is called constant speed control(G97 modal); other is the tangent speed of tool relative to the outer circle of workpiece defined by S command, which is called constant surface speed control (G96 modal), and the spindle speed is changed along with the absolute coordinates value of X absolute coordinates in programming path when cutting feed is executed in the constant surface speed.

3.2.2 Spindle override

When the spindle speed analog voltage control is valid, the spindle actual speed can be tuned real time by the spindle override and is limited by max spindle speed of current gear after the spindle override is tuned, and it also limited by limited values of max. and min. spindle speed in constant surface speed control mode.

The system supplies 8 steps for spindle override (50%～120% increment of 10%). The actual steps and tune of spindle override are defined by PLC ladder and introductions from machine manufacturer should be referred when using it. Refer to the following functions of GSK988T standard PLC ladder.

The spindle actual speed specified by GSK988T standard PLC ladder can be tuned real time by the spindle override tune key at 8 steps in 50%～120% and it is not reserved when the spindle override is switched off. Refer to the operations of spindle override in Ⅱ OPERATION.

3.3 Tool Function

3.3.1 Tool offset

T functions of GSK988T: automatic tool change and executing tool offset. Control logic of automatic tool change is executed by PLC and tool offset is executed by NC.

Command format:

T □□ ○○

Tool offset number（the leading zero cannot be omitted）

Target tool number（the leading zero cannot be omitted）

Command function: The automatic tool post rotates to the target tool number and the tool offset of tool offset number commanded is executed. The tool offset number can be the same as the tool number, and also cannot be the same as it, namely, one tool can corresponds to many tool offset numbers. After executing tool offset and then T□□00, the system reversely offset the current tool offset and the system its operation mode from the executed tool length compensation into the non-compensation, which course is called the canceling tool offset, called canceling tool compensation. When the system is switched on, the tool offset number and the tool offset number displayed by T command is the state before the system is switched off, the tool offset number is in the cancelling state(i.e. 00 state).No. 3032 sets T code digit, and No.5002 Bit 0(LD1) sets the digit of tool offset number

Toolsetting is executed to gain the position offset data before machining (called tool offset), and the system automatically executes the tool offset after executing T command when programs are running. Only edit programs for each tool according to part drawing instead of relative position of each tool in the machine coordinate system. If there is error caused by the wearing of tool, directly

modify the tool offset according to the dimension offset.

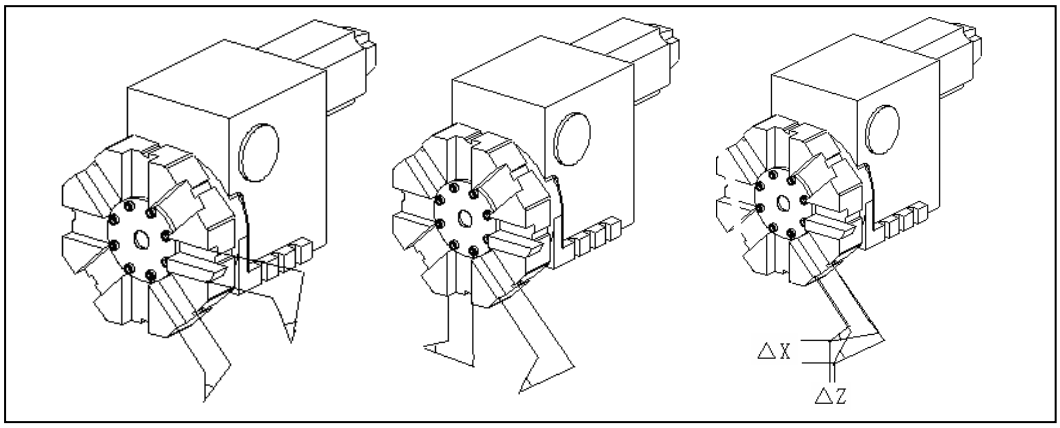


Fig.3-5 Tool offset

The tool offset is used for the programming. The offset corresponding to the tool offset number in T command is added or subtracted on the end point of each block. X tool offset in diameter or radius is set by No.5004 Bit1(ORC). For X tool offset in diameter or radius, the external diameter is changed along with diameter or radius when the tool length compensation is changed.

Example: When the state parameter No.5004 Bit1 is set to 0 and X tool length compensation value is 10mm, No.5004 Bit1 is set to 1 and X tool length compensation value is 10mm the diameter of workpiece external diameter is 20mm.

Fig. 3-6 is to create, execute and cancel the tool offset in movement mode.

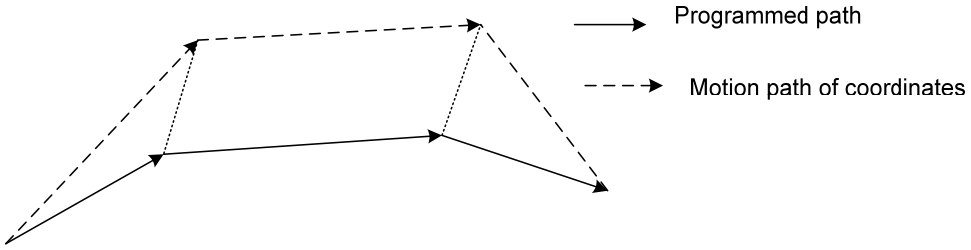


Fig. 3-6 Creation, execution and cancellation of tool length

G01 X100 Z100 T0101； （Block 1, start to execute the tool offset）
G01 W150； （Block 2, tool offset）
G01 X50 Z300 T0100； （Block 3, canceling tool offset）

There are two methods to execute the tool offset(they are set by No.5002 Bit4(LGT)):

（1）The tool length compensation is executed by the tool traversing;

（2）The tool length compensation is executed by modifying the coordinates;

Example:

Table 3-1

Tool offset number	X	Z
00	0.000	0.000
01	0.000	0.000
02	12.000	-23.000
03	24.560	13.452

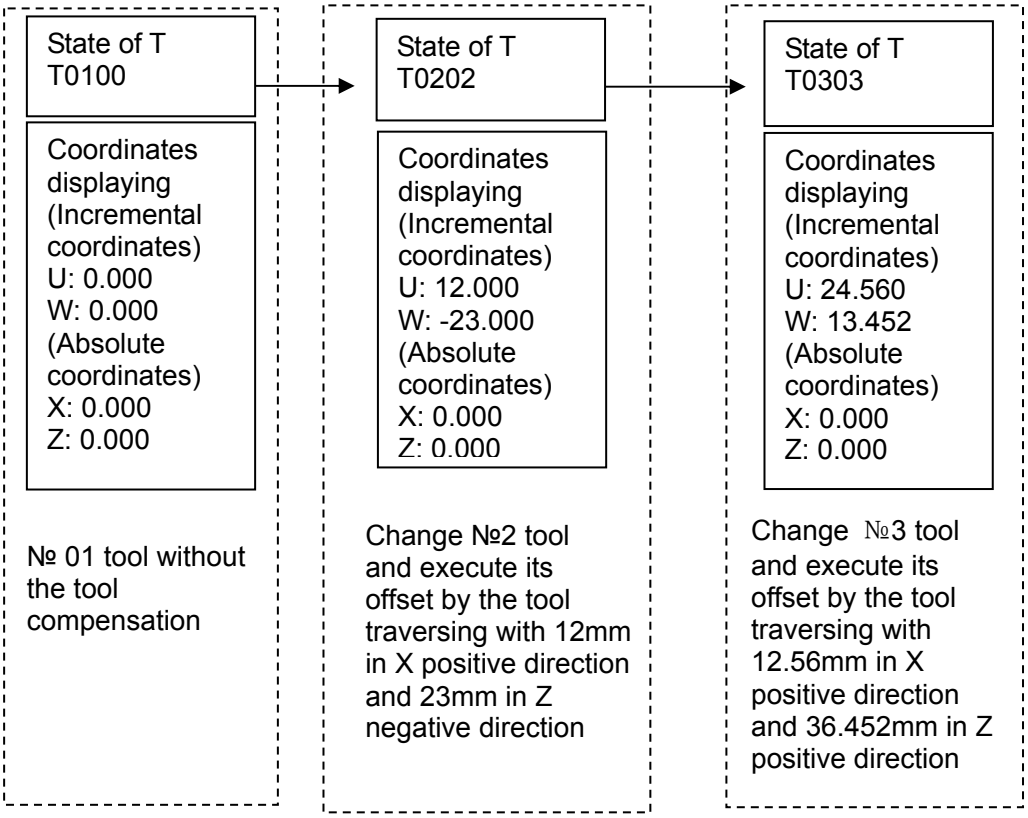

State of T
T0100
Coordinates displaying (Incremental coordinates)
U: 0.000
W: 0.000
(Absolute coordinates)
X: 0.000
Z: 0.000
№ 01 tool without the tool compensation
State of T
T0202
Coordinates displaying (Incremental coordinates)
U: 12.000
W: -23.000
(Absolute coordinates)
X: 0.000
Z: 0.000
Change №2 tool and execute its offset by the tool traversing with 12mm in X positive direction and 23mm in Z negative direction
State of T
T0303
Coordinates displaying (Incremental coordinates)
U: 24.560
W: 13.452
(Absolute coordinates)
X: 0.000
Z: 0.000
Change №3 tool and execute its offset by the tool traversing with 12.56mm in X positive direction and 36.452mm in Z positive direction

Fig. 3-7 Tool traversing mode to execute the tool offset

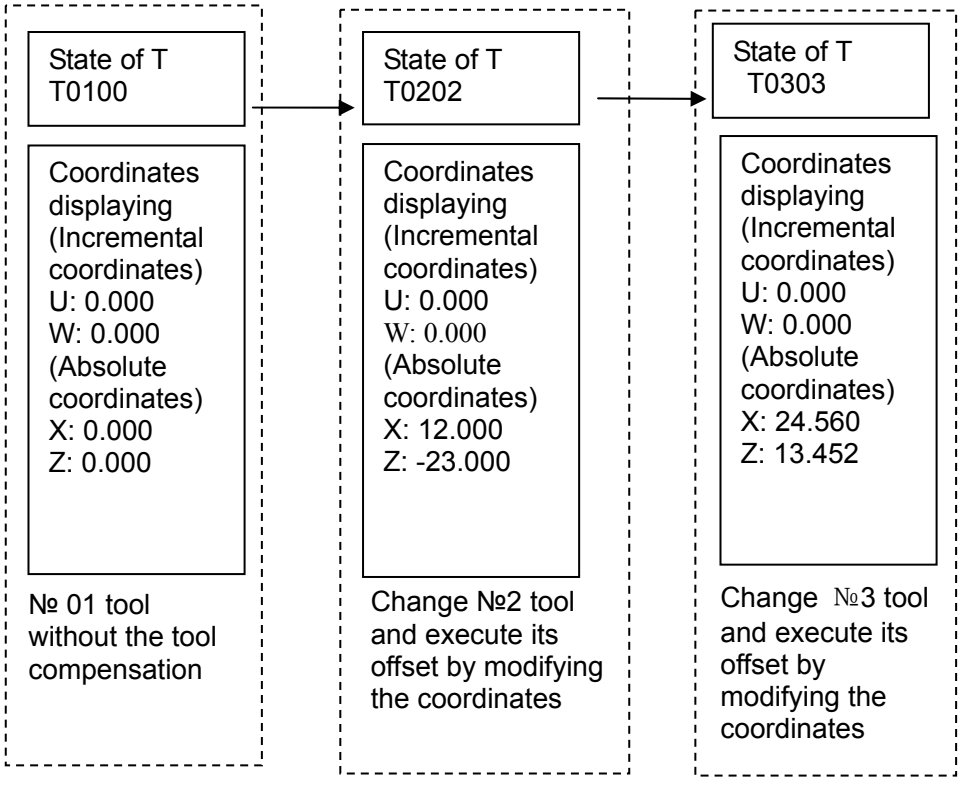


Fig. 3-8 Modifying the coordinates mode to execute the tool offset

When T command and the motion command are in the same block, they are executed simultaneously, in executing tool change, the system executes by adding the current tool offset to coordinates of motion command and whether the cutting feedrate or the rapid traverse speed is

defined by the motion command.

Note 1: In tool traversing compensation mode, when the system executes the tool offset, NO. 5002 Bit6 sets the valid method of the tool offset:

LWM=0: it is valid in the block of T code
LWM=1: it is valid in the axis movement block

Note 2: No.5001 Bit6（EVO）sets the valid method of modifying tool wear tool when the system executes the program:

EVO =0: it is valid in the next specified T code
EVO =1: it is valid in the next buffer block

Note: After the tool wear value is modified, the system executes the wear value in the tool traversing method to avoid the too big wear value.

Note 3: It is suggested that the program should be complied according to the normative method, that is, the tool change is executed in the safe position and then the absolute value command is used to position to the starting point of the machining.

Note 4: In the coordinate offset compensation mode, when the system executes the tool offset and executes T function command instead of movement command, it uses G50 to set the coordinate system, the displayed absolute coordinate value is that the coordinate value set by G50 adds or subtracts the unexecuted tool compensation value.

Note 5: T command can use the leading zero. When T00□□ is commanded or only tool offset number is commanded in the program and the current tool number is not changed, the system only correspondingly modifies the current tool offset value.

Note 6: After executing the miscellaneous function lock is valid, the system does not execute the tool change when T command is executed but the tool offset is executed. When using the miscellaneous function lock checks the program function, it is executed in the safety position.

3.3.2 Tool Life Management

When the tool life management is used, TLF (No. 8132#0) is set to “1”.

The tools are divided into many groups. Each group specifies its corresponding tool life (the used time or used times), and the used time or times are accumulated. When the current tool life arrives, the next tool in the same group is selected according to the preset order. The system alarms when all tool life are used in the same group.

3.3.2.1 Tool Life Management Data

Set the used most group number in No. 6813. Bit0 and Bit1 (GS1 and GS2) of No.6800 set the actual group number and the most tool in each group.

GS2	GS1	Group number	Tool number
0	0	1/8 of 1～most group number (N0.6813)	1～16
0	1	1/4 of 1～most group number (N0.6813)	1～8
1	0	1/2 of 1～most group number (N0.6813)	1～4
1	1	1～most group number (N0.6813)	1～2

Note 1: After the above No. 6813 or No.6800 Bit0 and Bit1 (GS1 and GS2) are changed, inputting programs of tool life data resets the tool life;

Note 2: The same tool number appears in any time in programs of tool life data;
Note 3: T command is consisted of the tool selection number and the tool offset number;
Note 4: No.5002 Bit 0 must be set to 0 when the tool life management function is used.

3.3.2.2 Tool Life Timing/Counting

When the tool life counting method in the input program of tool life data is not specified, LTM (No.6800#2) value is set to confirm the time or times to specify the tool life.

1. Specify tool life in used time

Specifty the unit of tool life according to FGL (#6805.1) (0: 1m; 1: 0.1s).

When the tool group command （T○○９９）is specified, the tool which life does not reach is selected and the tool life management of the selected tool is done. (time interval is set by FCO(（No.6805#0)（0 : the interval is 1 for 1s；0.1s interval control is changed by override）. The time of single block stop, feed hold, rapid traverse, pause, machine lock and interlock is not counted into the current used tool life.

The tool life is up to 4300 minutes, the set most life is 4300m or 2580000(01s) according to #6805.1.

2. Specify tool life according to used times

When the tool group command (T□□９９) is specified, the tool which life does not reach and 1 is added to the life of the selected tool. But, when the tool life counting is not specified and M command is activated, the new tool selection and counting are done after the system runs from the reset state to start state in Auto mode and it executes the No. 1 tool group command and the tool change command

The tool life is up to 65535 times.

Note: Even if the same tool group number is specified many time in one program the used times cannot be accumulated and the new tool is not also selected.

3.3.2.3 Tool Life Counting and Activating M Command

When the life counting is specified by times, the life reaches when the tool life counting is specified and M command is activated.

The tool change signal does not output even if there is only one tool in the tool group. In the tool group command (T command) which tool life is counted and M command is activated, the tool which life does not reach in the specified group, 1 is added to the tool life counter.

Tool life counting and activating M command are specified by No. 6811.

3.3.2.4 Tool Life Management Command in Machining Programs

The tool life is used in machining programs, and T command specifies the tool group according to the following format.

Command format:

……

T□□99； end the tool life counting in the previous group, use the tool which life does not reach in Group □□ and output T signal, count the tool life of tools in Grou□□.

……

T□□88； end the tool life management of Group □□, cancel the tool offset which is being used, and T signal for the tool number is output

……

M02（M30）； end of machining program ；

Command function:

Machining is executed according to the specified group and the tool life management is executed

Example: bit number of imaginary offset number is 2	
T0199； ： ： ： ：	Select the tool which life has not reached in Group 1 （imagine that T1001 is selected, the tool number is 10 and the offset number is 01.） Select tool life count in Group 1 （execute life count of tool number 10）
T0188； ： ： ： ：	Cancel tool post offset which is being used in Group 1 （the tool being used is **T1001**, the tool number is 10 and the offset number is 00. ）
T0299； ： ： ： ：	Select the tool which life has not reached in Group 2 imagine that T2002 is selected, the tool number is 20 and the offset number is 02.） Select tool life count in Group 2 （execute life count of tool number 20）
T0299； ： ： ： ：	When tools being used in Group 2 command many offset numbers, the next offset number is selected. （When the tool number has T2002 and T2003, T2003 is selected, the tool number is 20 and the offset number is 03.）
T0301； ： ： ： ：	The tool life count function ends in Group 2, the tool number is 03 and the offset number is 01.

Note:
T□□99 is not commanded before T□□88, an alarm occurs.

3.3.2.5 Automatic Input of Tool Life Data

G10/G11 is used to input the tool life management data and its format is shown below:

（1）Delete data in all groups when the system logs in:

Format	Symbol explanation
G10 L3； **P- L-**； **T-**； **T-**； . **P- L-**； **T-**； **T-**； . **G11**； **M02(M30)**；	**G10 L3**:delete all groups when the system logs in. **P-**：group number **L-**：tool life value **T-**：tool number and tool offset number **G11**：log-in ends

After the system deletes all logged tool life management data, the system logs in programmed tool life management data.

（2）Change tool life management data

Format	Symbol explanation
G10 L3 P1；	**G10 L3 P1**: group data change starts

P- L-; **T-;** **T-;** . **P- L-;** **T-;** **T-;** . **G11;** **M02(M30);**	**P-**：group number **L-**：tool life value **T-**：tool number and tool offset number **G11**：log-in ends

The system can set the tool life management data in the unlogged tool life mangagement data group or change the logged tool life management data.

（3）Delete tool life management data ：

Format	Symbol explanation
G10 L3 P2; **P- ;** **P- ;** **P- ;** **P- ;** . **G11;** **M02(M30);**	**G10 L3 P2**:group data deletion starts **P-**：group number **G11**：deletion ends

（4）Set counting type of tool life group

Format	Symbol explanation
G10 L3 ; **（G10 L3 P1）;** **P- L- Q-;** **T-;** **T-;** . **G11;** **M02(M30);**	**Q**:life count type （1：times, 2：time）

Note: When Q command is omitted, life count type is set by setting value of parameter LTM (No.6800#2).

3.3.2.6 Processing after tool life ending

The tool change signal is output when the tool life counting is done and the last tool life has reached in the group. When the life counting is specified by time and the last tool life has reached in the group, the tool change signal is output immediately. When the number of times is specified, the last tool life has reached in the group, the CNC resets by M02 or M30 or the tool life counting is commanded and M command is activated, the tool change signal is output immediately.

When LFI (No.6804#6) is set to 1, the invalid signal LFCIV can switch the life counting to be valid or invalid.

When the invalid signal LFCIV of tool life counting is set to 1 and LFCIF in the invalid tool life counting becomes 1, the life counting is valid.

When the invalid signal LFCIV of tool life counting is set to 0 and LFCIF in the invalid tool life

counting becomes 0, the life counting is valid.

Chapter IV Tool Nose Radius Compensation

4.1 Application

4.1.1 Overview

Part program is compiled generally for one point of tool according to a workpiece contour. The point is generally regarded as the tool nose A point in an imaginary state (there is no imaginary tool nose point in fact and the tool nose radius can be omitted when using the imaginary tool nose point to program) or as the center point of tool nose arc (as Fig. 4-1). Its nose of turning tool is not the imaginary point but one arc owing to the processing and other requirement in the practical machining. There is an error between the actual cutting point and the desired cutting point, which will cause the over- or under-cutting affecting the part precision. So a tool nose radius compensation is needed in machining to improve the part precision.

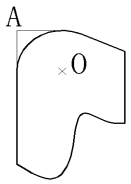


Fig. 4-1 Tool

B tool compensation is defined that a workpiece contour path is offset one tool nose radius, which cause there is excessive cutting at an intersection of two programs because of executing motion path of next after completing the previous block.

To avoid the above-mentioned ones, the system uses C tool compensation method (namely, tool nose radius compensation). The system will read the next block instead of executing it immediately after reading a block in C tool compensation method, and count corresponding motion path according to intersection of blocks. Contour can be compensated precisely because reading two blocks are pretreated as Fig.4-2.

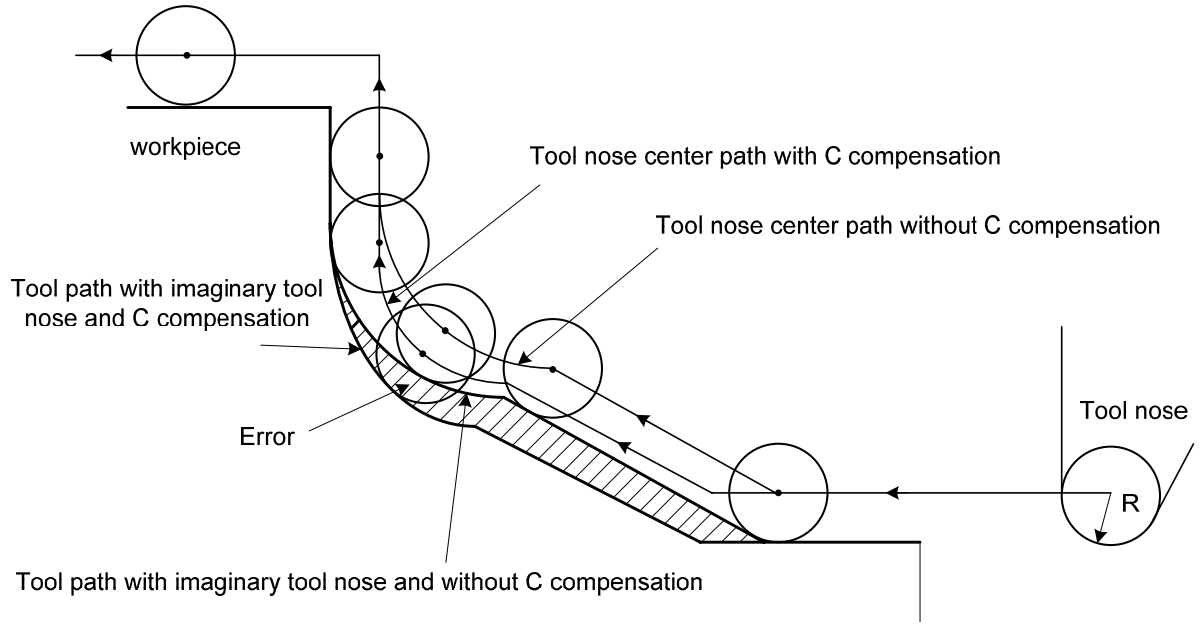
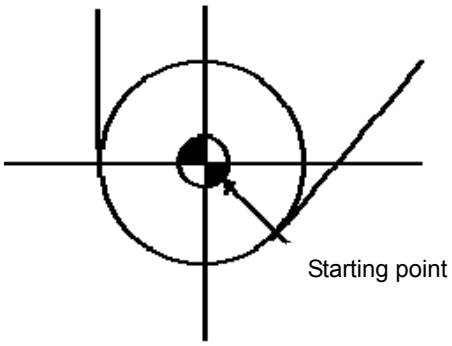

workpiece
Tool nose center path with C compensation
Tool nose center path without C compensation
Tool path with imaginary tool nose and C compensation
Error
Tool nose
R
Tool path with imaginary tool nose and without C compensation

Fig. 4-2

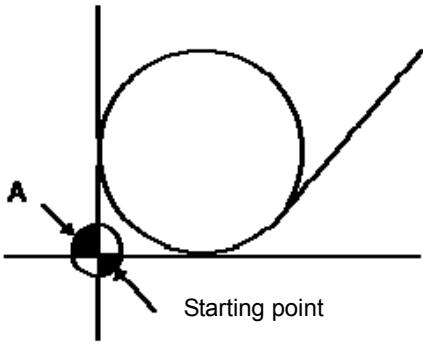
4.1.2 Imaginary tool nose direction

Suppose that it is generally difficult to set the tool nose radius center on the initial position as Fig. 4-3; suppose that it is easily set the tool nose on it as Fig. 4-4; The tool nose radius can be omitted in programming. Fig. 4-5 and Fig.4-6 correspond separately to the tool paths of tool nose center programming and imaginary tool nose programming when tool nose radius is executed or not.



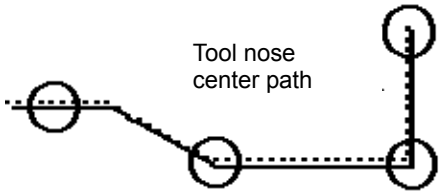
Programming with tool nose center

Fig. 4-3

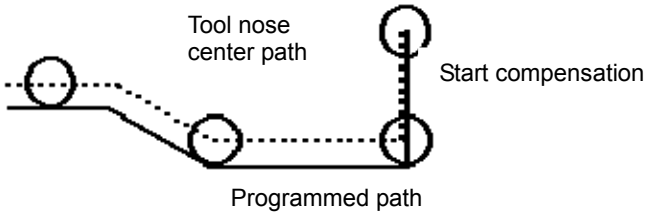


Programming with imaginary tool nose

Fig. 4-4

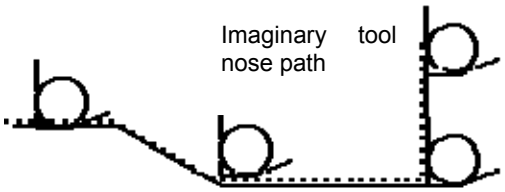


Tool nose path is the same as programming path without using tool nose radius compensation



Finishing when using tool nose radius compensation

Fig. 4-5 Tool path in tool nose center programming



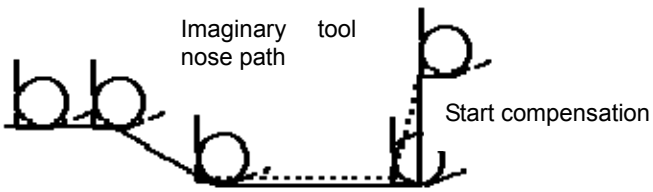
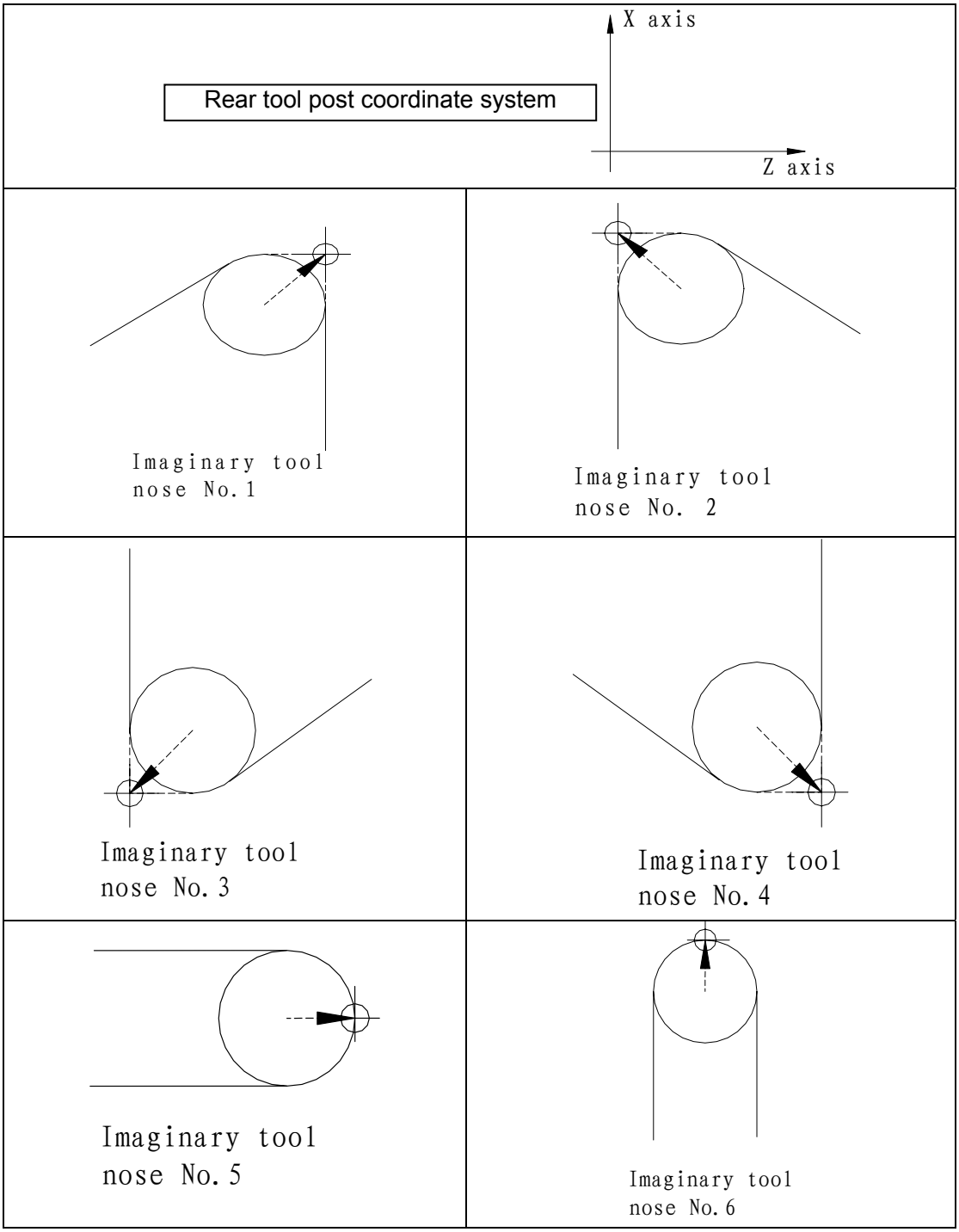


Fig. 4-6 Tool path in imaginary tool nose programming

The tool is supposed to one point in programming but the actual cutting blade is not one ideal point owing to machining technology. Because the cutting blade is not one point but one circular, machining error is caused which can be deleted by tool nose circular radius compensation. In actual machining, suppose that there are different position relationship between tool nose point and tool nose circular center point, and so it must create correct its direction of imaginary tool nose.

From tool nose center to imaginary tool nose, set imaginary tool nose numbers according to tool

direction in cutting. Suppose there are 10（T0～T9） kinds of tool nose setting and 9 directions for position relationship. The tool nose directions are different in different coordinate system (rear tool post coordinate system and front tool post coordinate system) even if they are the same tool nose direction numbers as the following figures. In figures, it represents relationships between tool nose and starting point, and end point of arrowhead is the imaginary tool nose; T1～T8 in rear tool post coordinate system is as Fig. 4-7; T1～T8 in front tool post coordinate system is as Fig. 4-8. The tool nose center and starting point for T0 and T9 as Fig. 4-9.


X axis
Rear tool post coordinate system
Z axis
Imaginary tool nose No.1
Imaginary tool nose No. 2
Imaginary tool nose No. 3
Imaginary tool nose No. 4
Imaginary tool nose No. 5
Imaginary tool nose No. 6

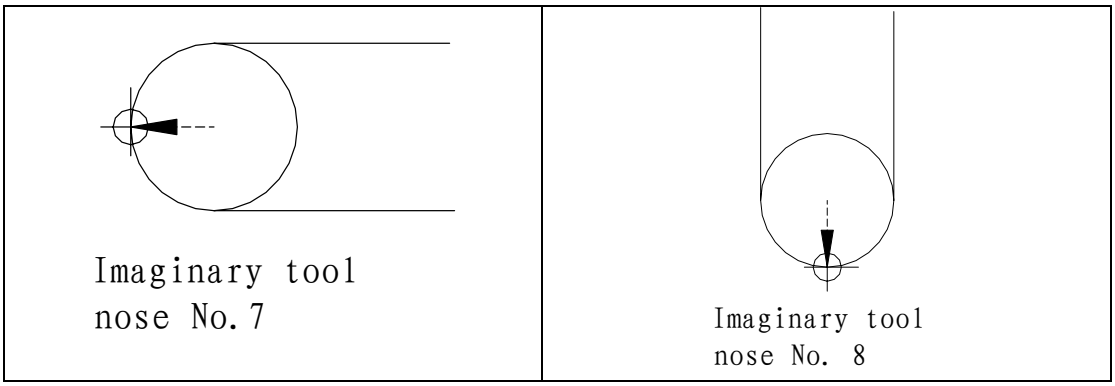
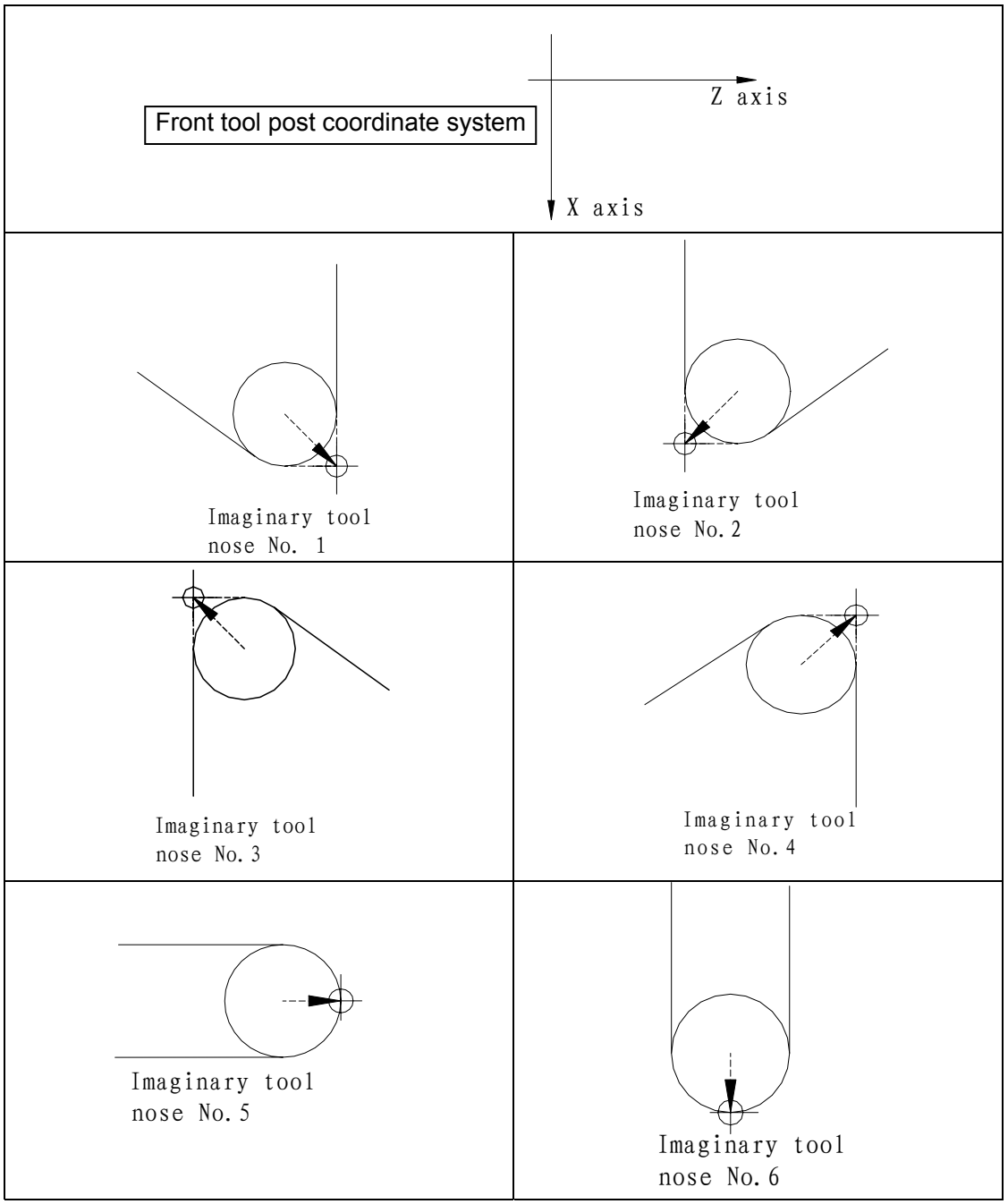


Fig. 4-7 Imaginary tool nose number in rear tool post coordinate system



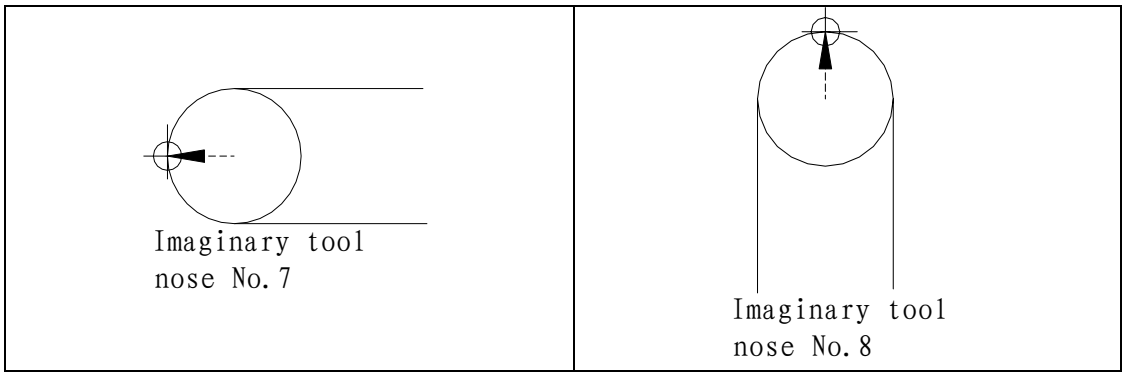


Fig. 4-8 Imaginary tool nose number in front tool post coordinate system

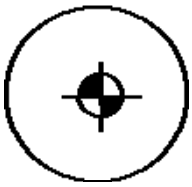


Fig. 4-9 Tool nose center on starting point

Note: The general imaginary tool nose direction 1～8 are used to G18 level, the imaginary tool nose 0 or 9 is used to G17 and G19 levels. The imaginary tool 0 or 9 used to G18 is valid, but the imaginary tool nose direction 1～8 are used to G17 and G19 levels, the system uses the nose 0 to execute the compensation.

4.1.3 Compensation value setting

Preset imaginary tool nose number and tool nose radius value for each tool before executing tool nose radius compensation. Set the tool nose radius compensation value in **"TOOL OFFSET&WEAR"** window (as Fig. 4-1), R is tool nose radius compensation value, T is imaginary tool nose number, and the radius compensation value is the sum of offset radius and wear radius.

Table 4-1 Display window of system tool nose radius compensation value

Tool offset No.		X	Z	…	R	T
001	Offset	0.000	0.000	…	0.380	3
	Wear	0.000	0.000	…	0.000	
002	Offset	10.000	10.000	…	0.250	3
	Wear	0.020	0.040	…	0.000	
003	Offset	14.000	15.000	…	1.200	3
	Wear	1.020	0.123	…	0.000	
…	Offset	…	…	…	…	…
	Wear	…	…	…	…	
099	Offset	10.000	12.000	…	0.300	0
	Wear	0.050	0.058	…	0.000	

In toolsetting, the tool nose is also imaginary tool nose point of Tn (n=0~9) when taking Tn(n=0~9) as imaginary tool nose. For the same tool, offset value from standard point to tool nose radius center (imaginary tool nose is T3) is different with that of ones from standard point to imaginary tool nose(imaginary tool nose is T3) when T0 and T3 tool nose points are selected to toolsetting in rear tool post coordinate system, taking tool post center as standard point. It is easier to measure

distances from the standard point to the tool nose radius center than from the standard point to the imaginary tool nose, and so set the tool offset value by measuring distance from the standard point to the imaginary tool nose(tool nose direction of T3).

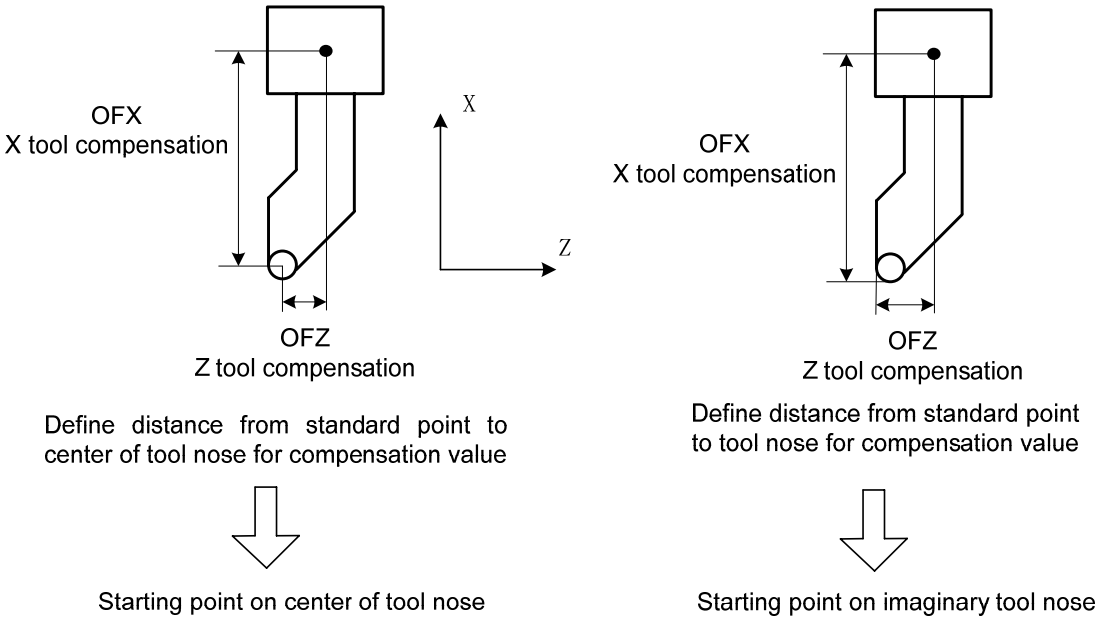


Fig. 4-10 Tool offset value of tool post center as benchmark

4.1.4 G40/G41/G42 command function

Taking the previous and the current position increment as the programmed path can cancel the tool compensation mode, and its direction is the compensation direction of the previous. When the system specifies（I, J），（I, K）or（J, K）, the vector defined by it can replace the current position increment to execute the count.

Command format:

$$\left\{\begin{matrix}G40\\G41\\G42\end{matrix}\right\}\left\{\begin{matrix}G00\\\\G01\end{matrix}\right\}\quad X\_\_\quad Z\_\_\quad T\_\_\ ;$$

In machining workpiece, the tool offset cannot easily compensate the precise workpiece because of the tool nose circle degree but the tool nose radius compensation function can automatically compensate the error.

G40 Xp__ Yp __ Zp__ I__ J__ K __

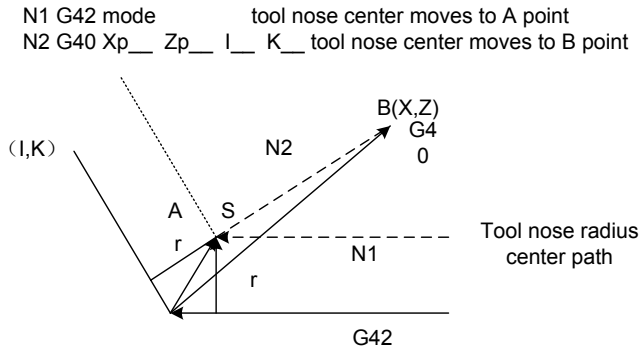


Fig. 4-11 G40 execution process

Command explanation:

Table 4-2

Commands	Function specifications	Remark
G40	Cancel the tool nose radius compensation	See Fig.4-11 and 4-12
G41	Tool nose radius left compensation is specified by G41 in rear tool post coordinate system and tool nose radius right compensation is specified by G41 in front tool post coordinate system	
G42	Tool nose radius right compensation is specified by G42 in rear tool post coordinate system and tool nose radius left compensation is specified by G42 in front tool post coordinate system	
Xp	X and its parallel axis	
Yp	Y and its parallel axis	
Zp	Z and its parallel axis	
I	X and the cancel vector (radius value) of its parallel axis	
J	Y and the cancel vector (radius value) of its parallel axis	
K	Z and the cancel vector (radius value) of its parallel axis	

4.1.5 Compensation direction

Specify its direction according to relative position between tool nose and workpiece when executing tool nose radius compensation as Fig. 4-12 and Fig.4-13.

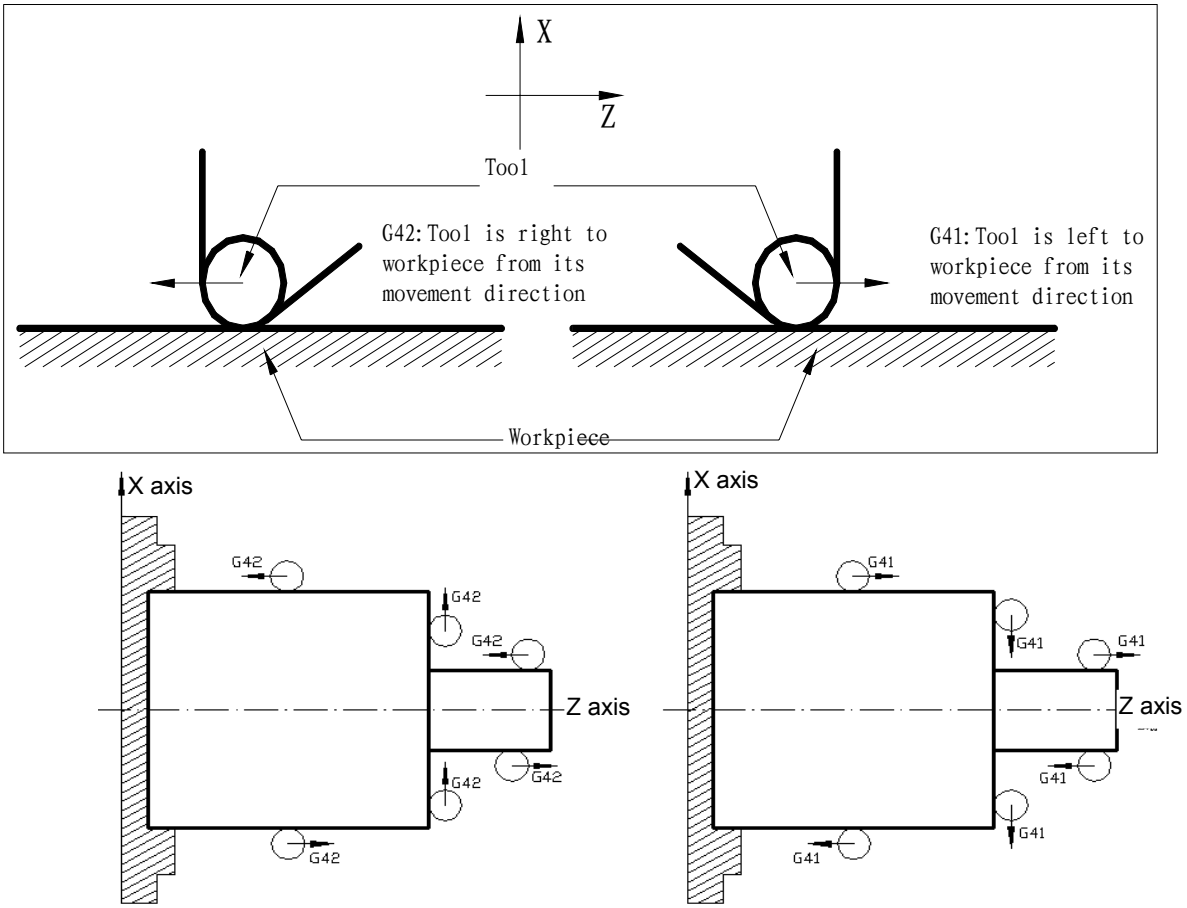


Fig. 4-12 Compensation direction of rear coordinate system

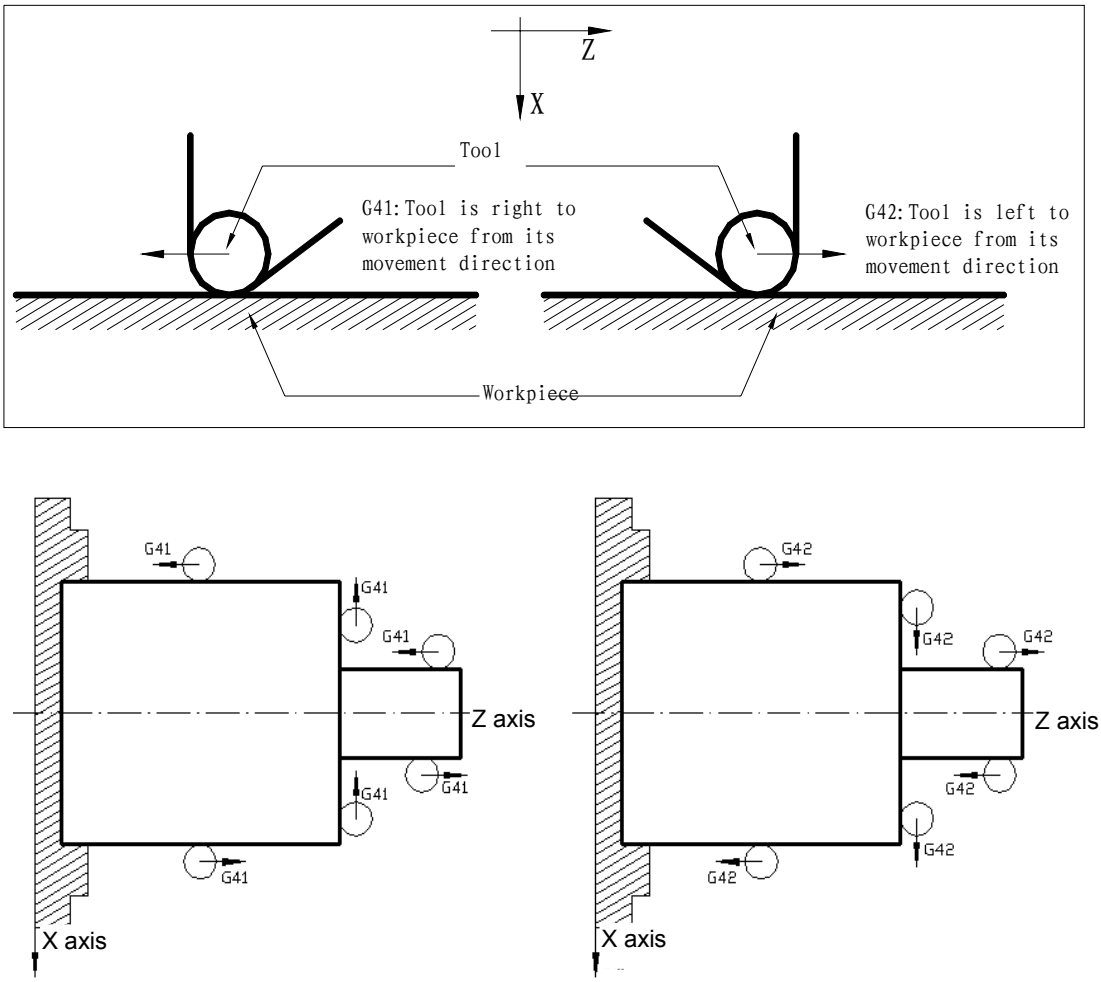


Fig. 4-13 Compensation direction of front coordinate system

4.1.6 Cautions

Note 1: In initial state, when the system is in the tool nose radius compensation cancel mode, and the offset compensation number is not 0 in G41 or G42, the system starts creating the tool nose radius compensation offset mode; when the offset compensation number is 0, G modal is the G40 state.

Note 2: In creating or cancelling tool compensation, the workpiece machining must not be executed, otherwise, it causes the overcut or undercut. The system takes the created first movement and the last movement command before being cancelled as the cutting command in normally machining workpiece.

Note 3: The tool does not create the offset and starts compensation in the next movement command when there is no movement command in creating the tool compensation. When there is no movement command in cancelling tool compensation, the tool does not create the offset and the system cancels the compensation vector in the next movement command.

Note 4: The next block to create the tool compensation block has the tool compensation cancel modal command, the system does not execute the tool compensation creation process, but at the moment, the modal command will change normally.

Note 5: The tool nose radius compensation creation and cancel only use G00 or G01 instead of G02 or G03. When they are specified, No.252 alarms.

Note 6: In tool nose radius compensation, the tool nose center moves to the end point of the last block and is vertical with the programmed path of the last when the system executes 3 or more than 3 blocks without movement command. At the moment, the overcut or undercut creates and the system should not machine the workpiece in the next block in programming. When 3 or more than 3 blocks without movement command following the movement command to create the tool nose radius compensation, the system does not create immediately the tool nose radius compensation but does it after the

non-movement command.

Note 7: The system does not execute the tool nose radius compensation in G50, G52, G32, G34, G92, G71, G72, G73, G74, G75, G76 and temporarily cancels the compensation mode. Before the system temporarily cancels the compensation execution and when the system modal is G02 or G03, No.262 alarms.

Note 8: In G40, for the inner or outer machining, the system moves to the intersection of two paths, and executes the tool nose radius compensation cancel here, and then moves to the target point after the cancel. When there is no intersection and the tool reaches the normal line position of the end point of the last block, the system cancels the tool nose radius compensation and then moves the target point after the cancel. At the moment, the overcut creates, the workpiece must not be machined.

Note 9: In tool nose radius compensation mode, the system must not be switched to other levels, otherwise, No.253 alarms.

Note 10: In tool nose radius compensation mode, the system cancels the tool compensation mode in RESET, M30 or M02 mode.

Note 11: In MDI mode, the system cannot execute the tool nose radius compensation creation and its cancel. When the system specifies the tool nose radius compensation command, it executes the command according to No.5008 Bit4(MCR). When the parameter is set to 1, the system alarms.

4.1.7 Application

Machine a workpiece in the front tool post coordinate system as Fig. 4-14. Tool number: T0101, tool nose radius R=2, imaginary tool nose number T=3.

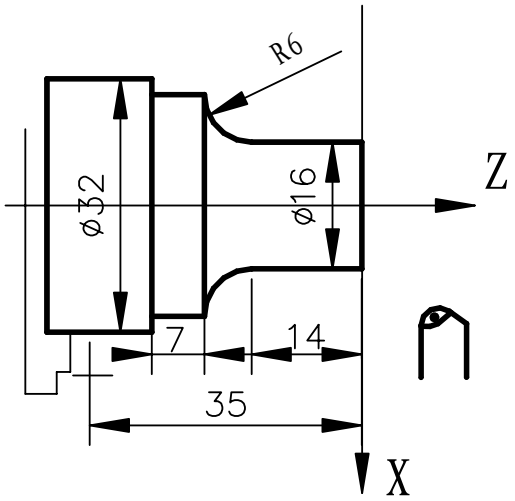


Fig. 4-14

Set the correct imaginary tool nose direction for executing the toolsetting in offset cancel mode, Set the tool nose radius R and imaginary tool nose direction in "***TOOL OFFSET & WEAR"*** window as following:

Table 4-3

No.	X	Z	Y	…	R	T
001				…	2.000	3
002	…	…	…	…	…	…
…	…	…	…	…	…	…
007	…	…	…	…	…	…

Program:

G00 X100 Z50 M3 T0101 S600; (Position, start spindle, tool change and execute tool compensation)

G42 G00 X0 Z3; (Set tool nose radius compensation)

```
G01 Z0 F300;                    (Start cutting)
X16;
Z-14 F200;
G02 X28 W-6 R6;
G01 W-7;
X32;
Z-35;
G40 G00 X90 Z40;                (Cancel tool nose radius compensation)
G00 X100 Z50 T0100;
M30;
```


4.2 Tool Nose Radius Compensation Offset Path

4.2.1 Inner and outer side

Inside is defined that an angle at intersection of two motion blocks is more than or equal to 180°; **Outside** is 0~180°.

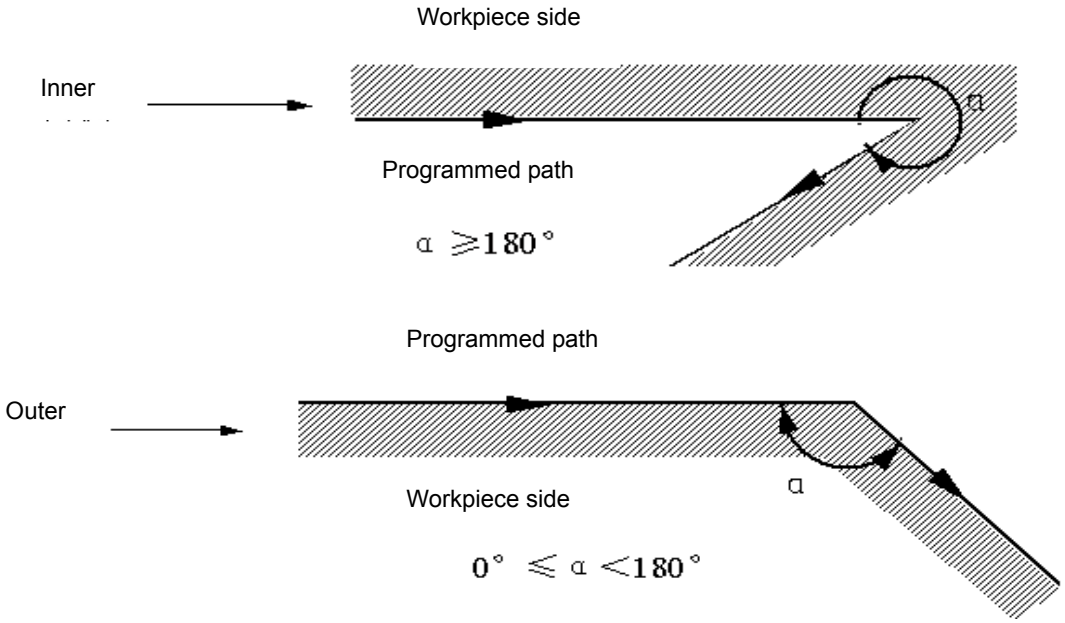


Fig. 4-15

4.2.2 Tool traversing when starting tool

3 steps to execute tool nose radius compensation: tool compensation creation, tool compensation execution and tool compensation canceling.

Tool traverse is called tool compensation creation (starting tool) from offset canceling to G41 or G42 execution.

Note: Meanings of S, L, C in the following figures are as follows:
S——Stop point of single block; L——linear; C——circular, R——tool radius compensation; α——angle between two blocks.

(a) Tool traversing inside along corner(α≥180°)

1) linear —linear

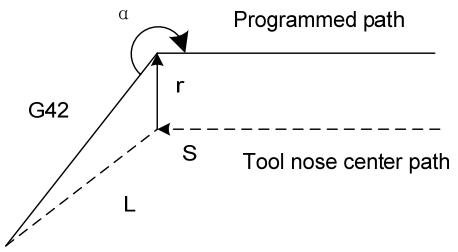


Fig.4-16 Linear —linear(starting tool inside)

2) linear —circular

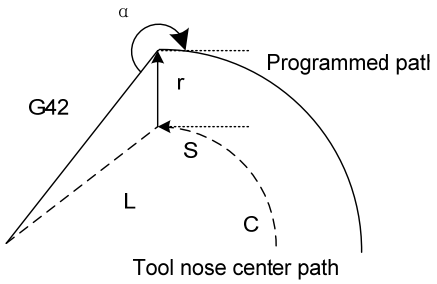


Fig. 4-17 Linear —circular（starting tool inside）

(b) Tool traversing inside along corner(180°＞α≥90°)

1) linear —linear

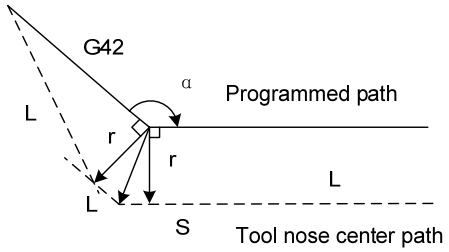


Fig.4-18 Linear —linear(starting tool outside)

2) linear —circular

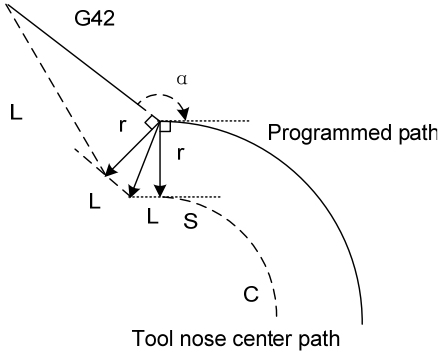


Fig.4-19 Linear—circular(starting tool outside)

(c) Tool traversing inside along corner (α＜90°)

1) linear —linear

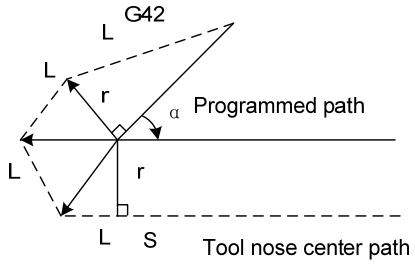


Fig.4-20 Linear —linear（starting tool outside）

2) linear —circular

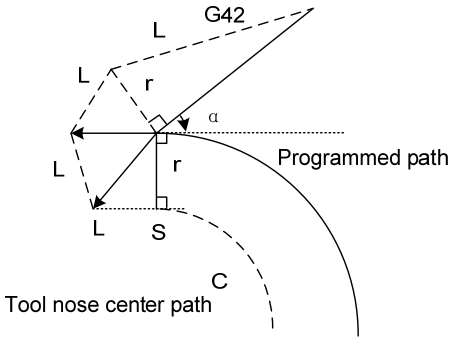


Fig. 4-21 Linear—circular (starting tool outside）

(d) Tool traversing inside along corner(α≦1°) , linear →linear

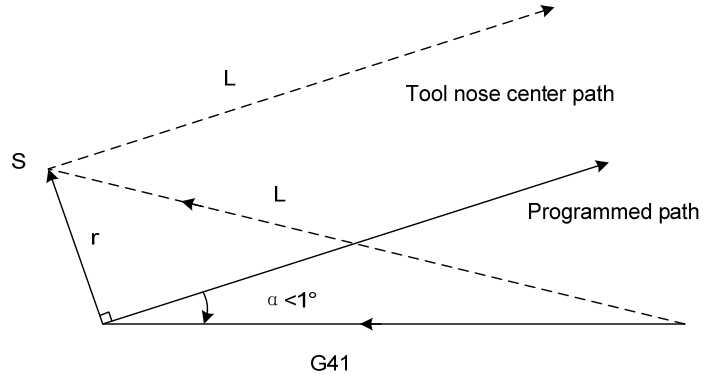


Fig. 4-22 Linear—linear（α＜1°, starting tool outside）

4.2.3 Tool traversing in Offset mode

Offset mode is called to ones after creating tool nose radius compensation and before canceling it.

- **Offset path without changing compensation direction in compensation mode**

(a) Tool traversing inside along corner(α≥180°)

1) linear —linear

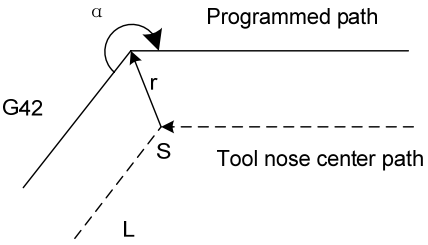


Fig. 4-23 linear—linear（moving inside）

2) linear —circular

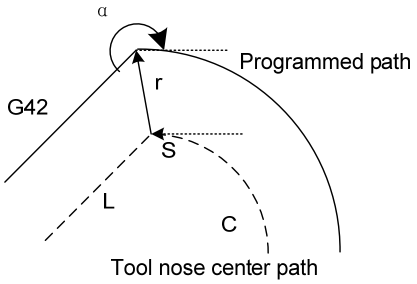


Fig. 4-24 linear—circular（moving inside）

3) circular—linear

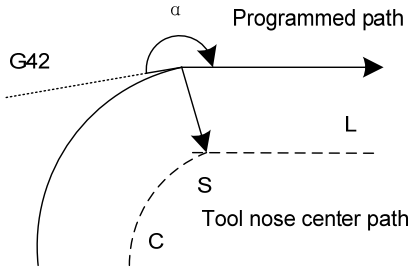


Fig. 4-25 Circular—linear（moving inside）

4) circular —circular

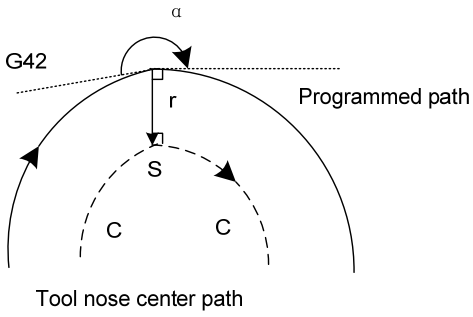


Fig. 4-26 Circular—circular（moving inside）

5）Machining inside (α＜1°) and zoom in the compensation vector

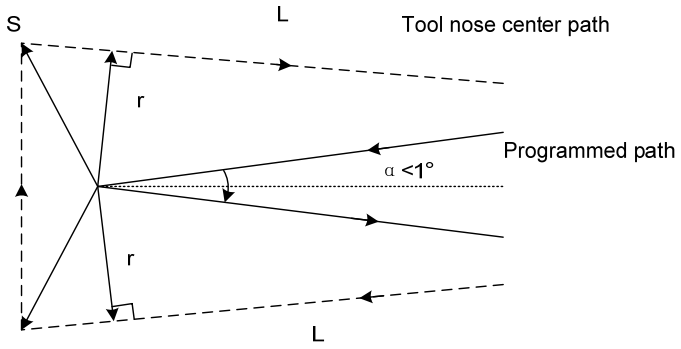


Fig. 4-27 Linear —linear（α＜1°, moving inside）

(b) Tool traversing outside along corner(180°＞α≥90°)

1) linear —linear

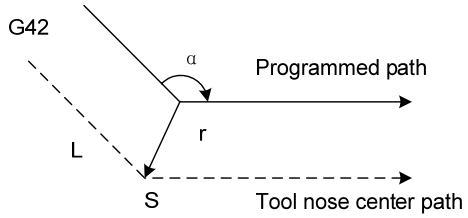


Fig. 4-28 Linear —linear（ moving outside）

2) linear —circular

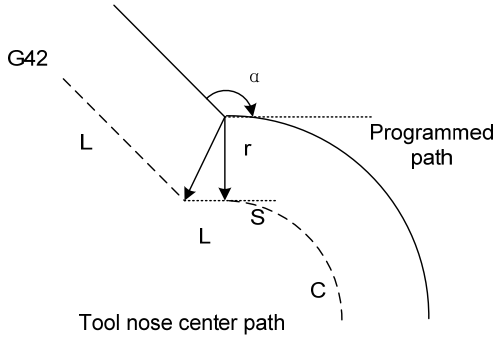


Fig. 29 Linear—circular（moving outside）

3) circular—linear

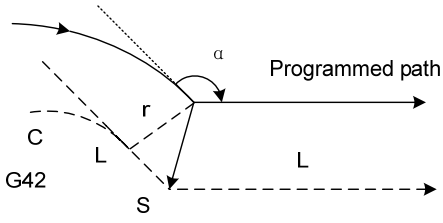


Fig. 4-30 circular—linear（ moving outside）

4)circular —circular

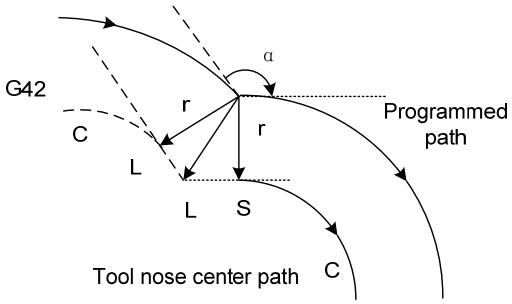
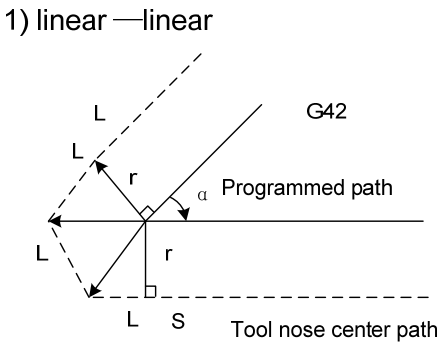


Fig. 4- 31 circular—circular（moving outside）

(c) Tool traversing outside along corner(α＜90°)



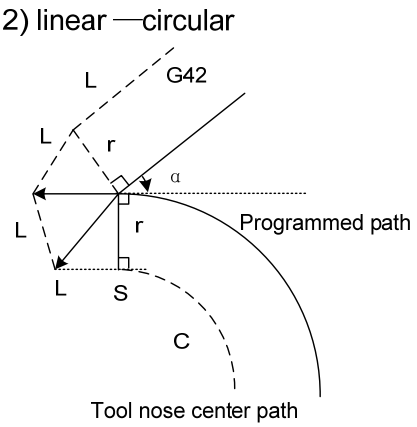
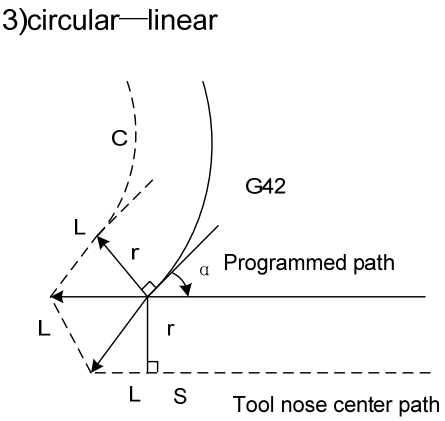


Fig. 4-32 Linear—Linea（moving outside） Fig. 4-33 Linear—circular（moving outside）



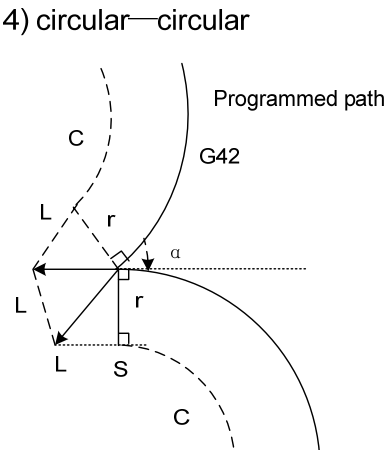


Fig.4-34 Circular—linear（moving outside ） Fig.4-35 Circular—circular（moving outside ）

(d) Special cutting

1) Without intersection

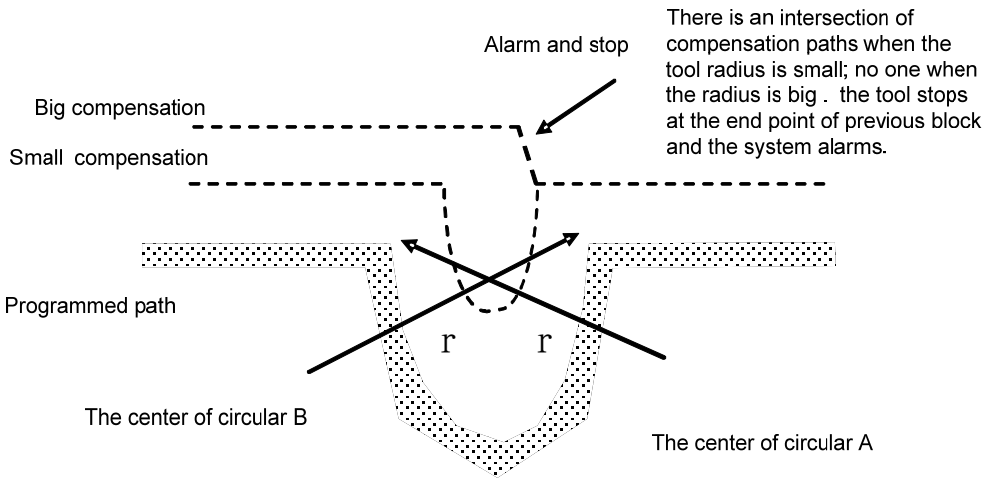

There is an intersection of compensation paths when the tool radius is small; no one when the radius is big . the tool stops at the end point of previous block and the system alarms.
Alarm and stop
Big compensation
Small compensation
Programmed path
r
r
The center of circular B
The center of circular A

Fig. 4-36 Paths without intersection after offset

2）Center point and starting point of circular being the same one

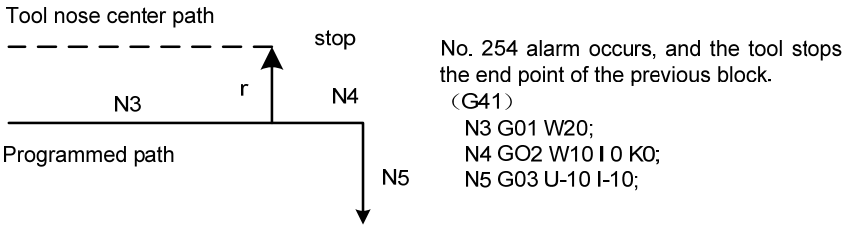


Fig. 4-37 Center point and starting point of circular being the same one

- **Offset path of compensation direction in compensation mode**

The compensation direction of tool nose radius is specified by G41 and G42 and the sign symbol is as follows:

Table 4-4

	Sign symbol of compensation value	
G Command	+	-
G41	Left compensation	Right compensation
G42	Right compensation	Left compensation

The compensation direction can be changed in compensation mode in special cutting, it cannot be changed at starting block and its following one. There is no inside and outside cutting when the system changes the compensation direction. The following compensation value is supposed to be positive.

1)linear—linear

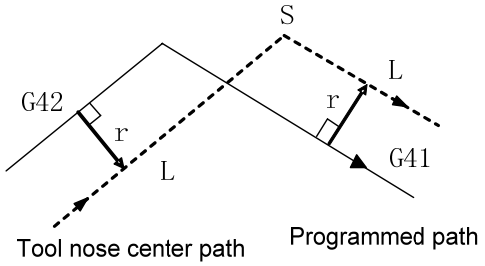


Fig. 4-38 Linear—linear （changing compensation direction）

2) linear—circular

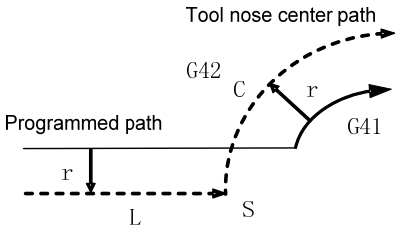


Fig. 4-39 Linear—circular （changing compensation direction）

3)circular—linear

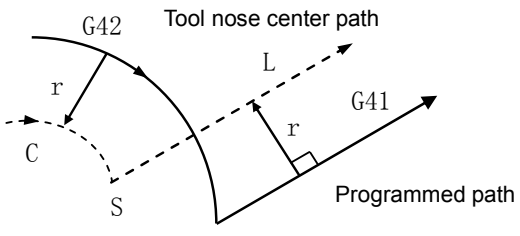


Fig. 4-40 circular—linear（changing compensation direction）

4) circular—circular

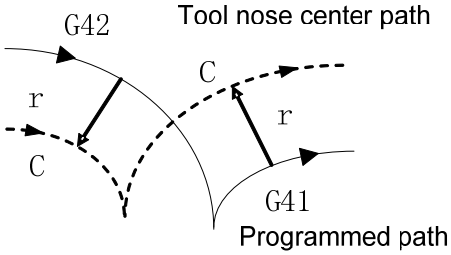


Fig. 4-41 circular—circular （changing compensation direction）

When the system executes G41 and G42 to change the offset direction between block A and B, a vector perpendicular to block B is created from its starting point.

i) Linear----Linear

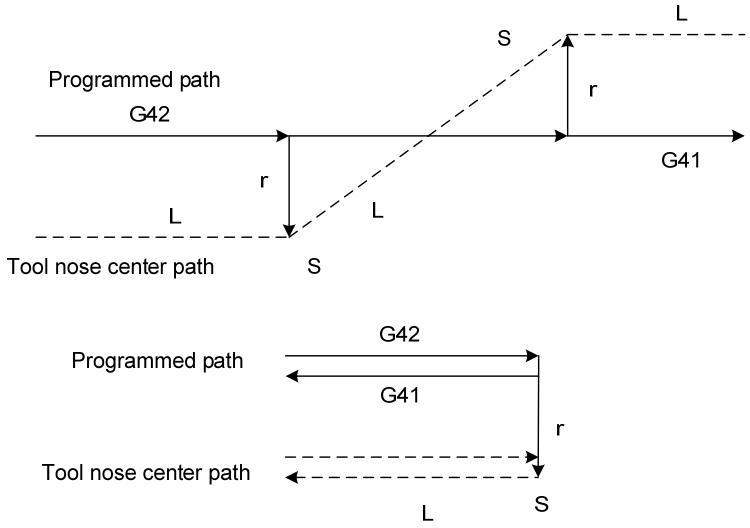


Fig. 4-42 Linear—linear, no intersection（changing compensation direction）

ii) Linear ---circular

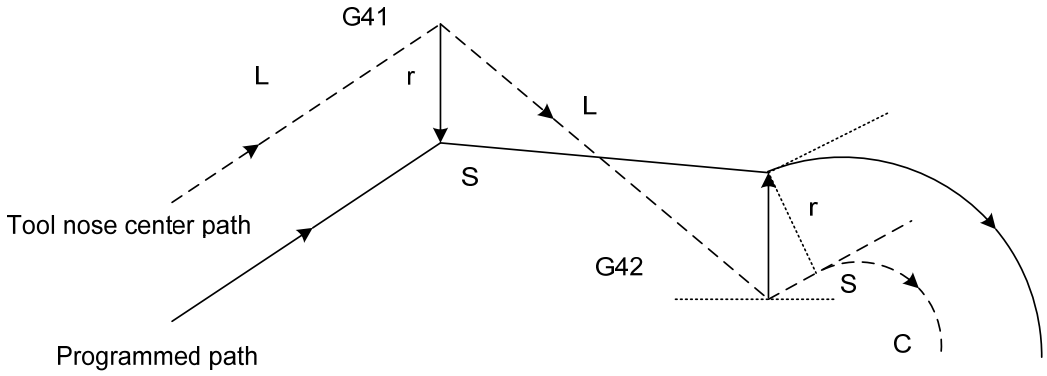


Fig. 4-43 Linear—circular without intersection（changing compensation direction）

iii) Circular-----circular

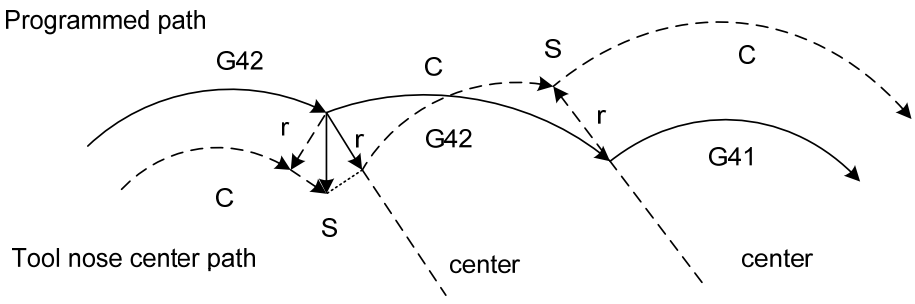


Fig. 4-44 Circular—circular without intersection (changing compensation direction)

4.2.4 Tool traversing in Offset canceling mode

In compensation mode, when the system executes G04, it enters the compensation canceling mode, which is defined to compensation canceling of block. The system cannot execute the circular command(G02 or G03) in canceling tool compensation mode, otherwise the system alarms and stops run.

(a) Tool traversing inside along corner(α≥180°)

1) linear —linear

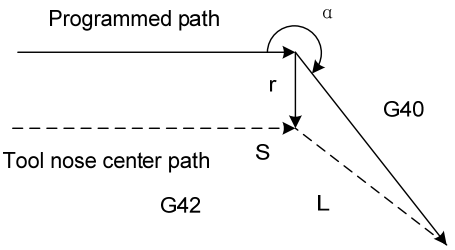


Fig. 4-45 Circular-linear (moving inner and canceling offset)

2) circular—linear

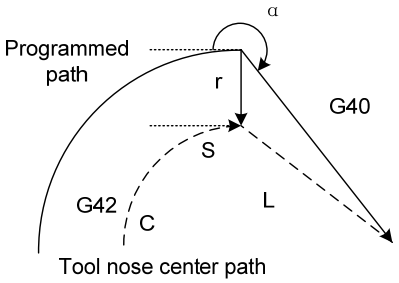


Fig. 4-46 Circular-linear (moving inner and canceling offset)

(b) Tool traversing outside along corner(180°＞α≥90°)

1) linear —linear

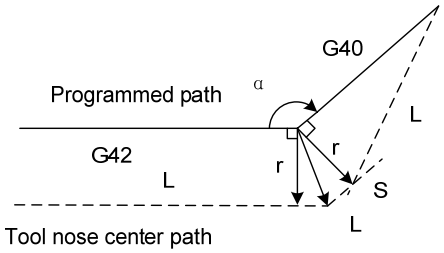


Fig. 4-47 Circular—linear（moving outside and canceling offset）

2) circular—linear

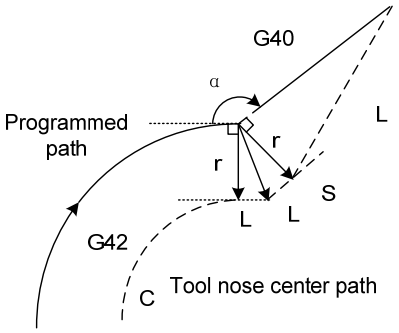


Fig. 4-48 Circular—linear（moving outside and canceling offset）

(c) Tool traversing outside along corner(α＜90°)

1) linear —linear

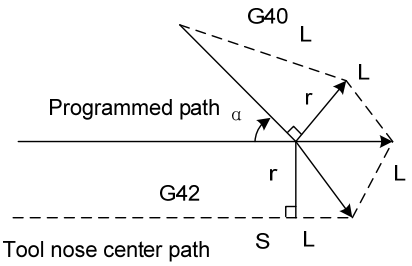


Fig. 4-49 Linear—linear (cutting outside and canceling offset)

2) circular —linear

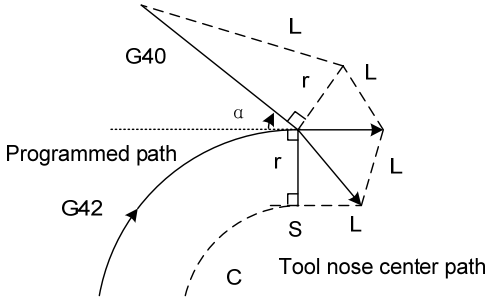


Fig. 4-50 Linear—linear (cutting outside and canceling offset)

(d) Tool traversing outside along corner(α＜1°) ; linear→ linear

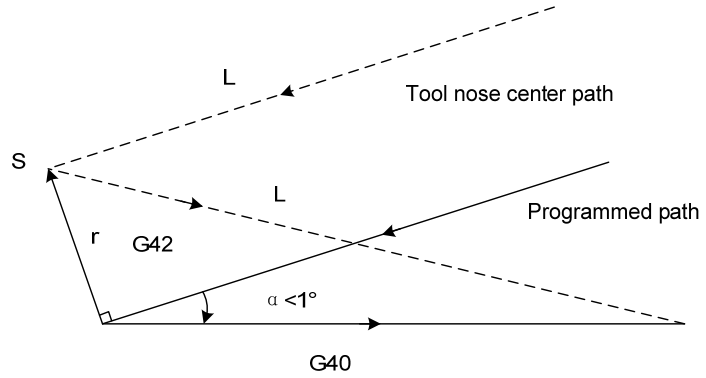


Fig. 4-51 Linear—linear (α＜1°cutting outside and canceling offset)

4.2.5 Tool interference check

"Interference" is defined that the tool cuts workpiece excessively and it can find out excessive cutting in advance, the interference check is executed even if the excessive cutting is not created, but the system cannot find out all tool interferences.

(1) Fundamental conditions

1) The tool path direction is different that of program path (angle is 90°~270°).

2) In machining arc, there is great difference the two angles(α＞180°), the one is between the starting point and the end point of the tool center path, and the other is between the starting point and the end point of the programmed path, or the system cuts the inner of the arc (α＞180°) , and the tool cannot pass the entrance, No.256 alarms.

Example: linear machining

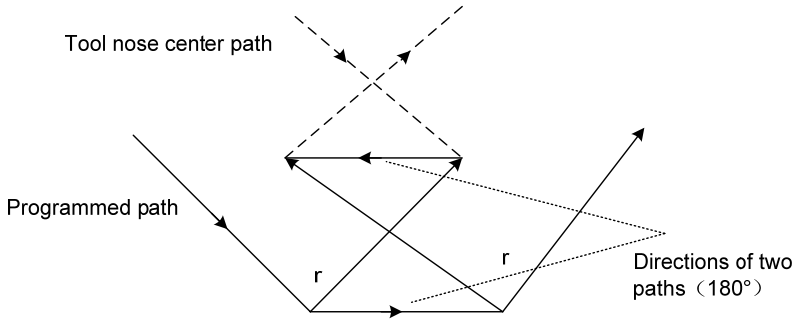


Fig. 4-52 Machining interference, No.257 alarm appears

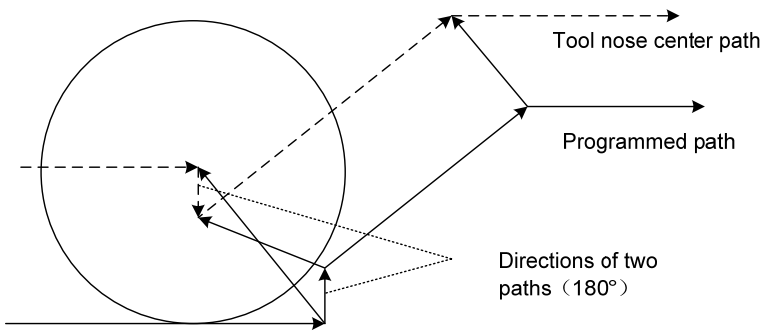


Fig. 4-53 Machining interference (2)

(2) Executing it without actual interference

1) Concave groove less than compensation value

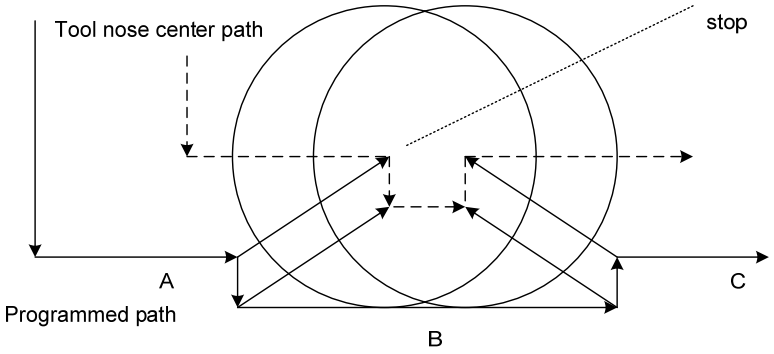


Fig. 4-54 Executing interference (1)

Directions of block B and tool nose radius compensation path are opposite without interference, the tools stops and the system alarms.

2) Concave channel less than compensation value

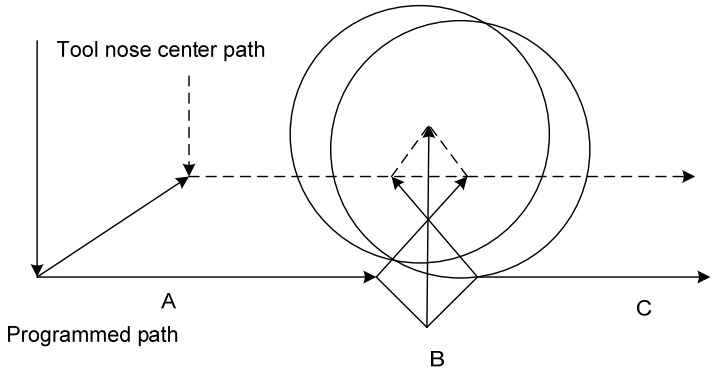


Fig. 4-55 Executing interference (2)

Directions of block B and tool nose radius compensation path are opposite without interference, the tools stops and No.257 alarms.

(3) Automatic interference vector clear

The system has the automatic interference vector clear function. For example, when the neighbor three blocks N10, N20, N30 execute the tool radius compensation, the section between N10 and N20 creates the vector V1, V2, V3 and V4, and the section between N20 and N30 creates V5, V6, V7, V8. The system executes the interference check to the last vectors in the above two group of vector, i.e. V4 and V5. V4 and V5 are ignored when there is the interference; the system checks V3 and V6, and they are ignored when there is the interference; the system does V2 and V7, and they are ignored when there is the interference. When the system executes the interference check to the last vectors V1 and V8, and there is the interference, they cannot be ignored, the tool stops movement and the system alarms. Based on the above process, the system executes the interference check, and has checked the vector which is not interfered, the followings are not check, and the tool runs according to the path of the first group vector which does not create the interference. When the last group of vector creates the vector, they cannot be ignored, the tool stops movement and No.257 alarms.

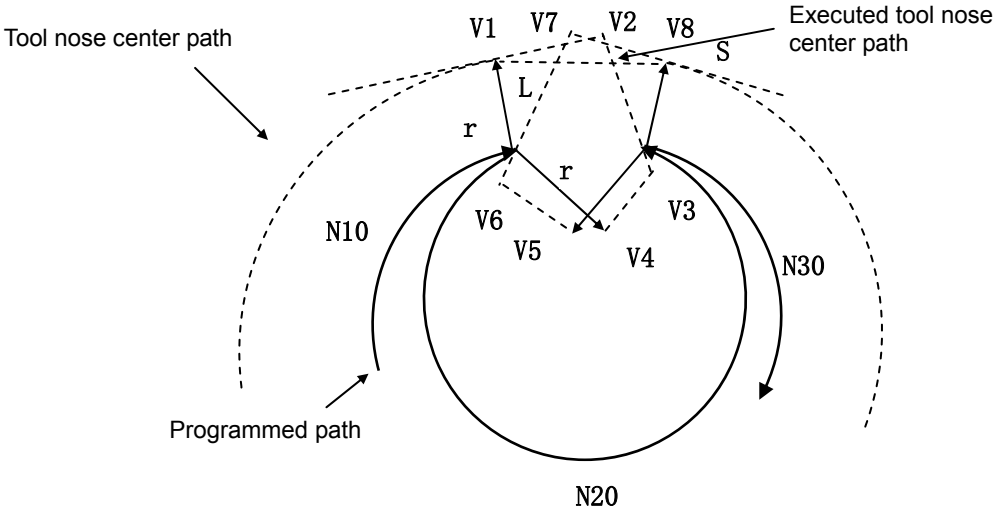


Fig. 4-56 interference vector clear

Note 1: NO.5008 Bit 0（CNI）can set whether the interference check is executed in tool nose radius compensation mode.

Note 2: NO.5008 Bit 1（CNC）can set whether the system alarms when the difference 90°-270° between the movement direction and offset direction.

Note 3: NO.5008 Bit 3（CNV）can set whether the system executes the interference check and the vector clear.

4.2.6 Commands for canceling compensation vector temporarily

In compensation mode, when the system specifies G28, G30, G50, G52, G32, G34, the fixed cycle, multi cycle, drilling cycle command, the compensation vector is cancelled temporarily and is automatically resumed after executing the commands. At the moment, the compensation is cancelled temporarily and the tool directly moves from intersection to a point for canceling compensation vector. The tool directly moves again to the intersection after the compensation mode is resumed.

- **Setting coordinate system in G50, G52**

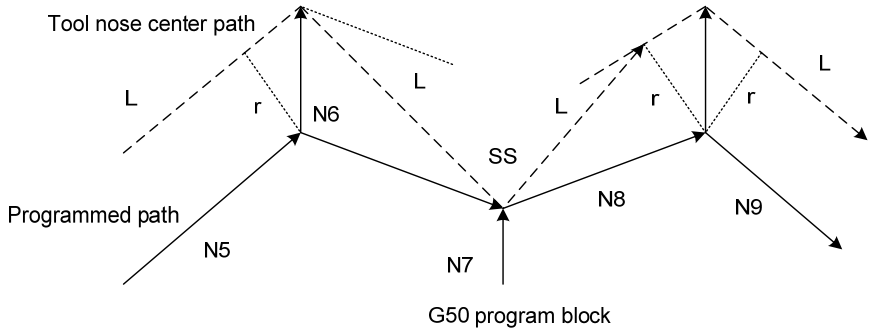


Fig. 4-57 Temporary compensation vector in G50, G52

Note: SS indicates a point at which the tool stops twice in Single mode.

- **Reference position automatic return G28, G30**

In compensation mode, the compensation is cancelled in a middle point and is automatically resumed after executing the reference position return in G28, G30.

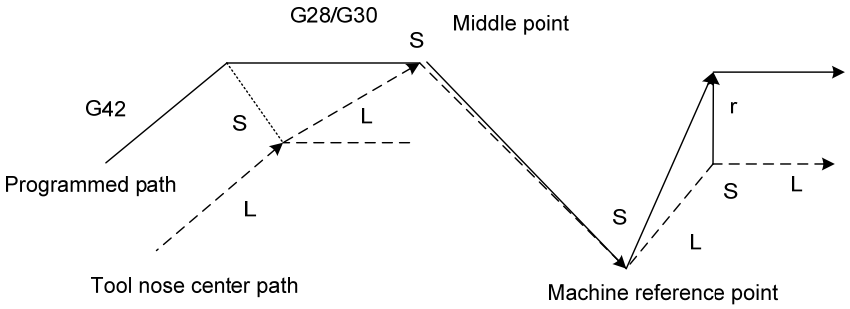


Fig. 4-58 Cancel compensation vector temporarily in G28

- **G53 automatic return to reference position**

In compensation mode, when G53 is commanded, the system creates the offset vector which is vertical with the tool motion direction before the end point of the last block. When the tool moves to G53 position, the compensation vector is cancelled. The compensation vector is automatically recovered when the system executes the next movement command.

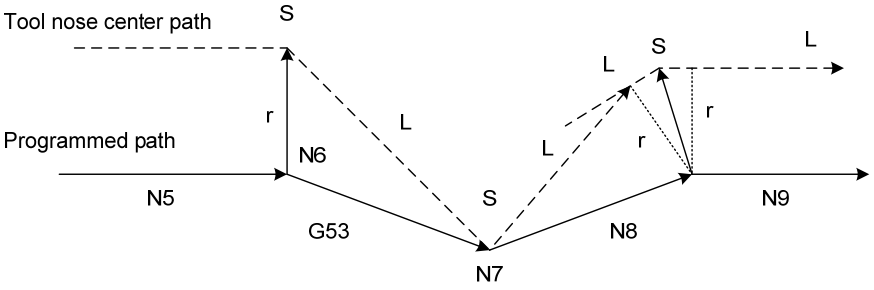


Fig. 4-59 G53 temporarily cancelling compensation vector

- **G71～G76 compound cycle; G92 fixed cycle, G84, G88 drilling cycle**

When executing G71～G76 , G92 fixed cycle, G84, G88 drilling cycle, the system does not execute the tool nose radius compensation and cancel it temporarily, and executes it in the next blocks of G00, G01, G70, CNC automatically recovers the compensation mode.

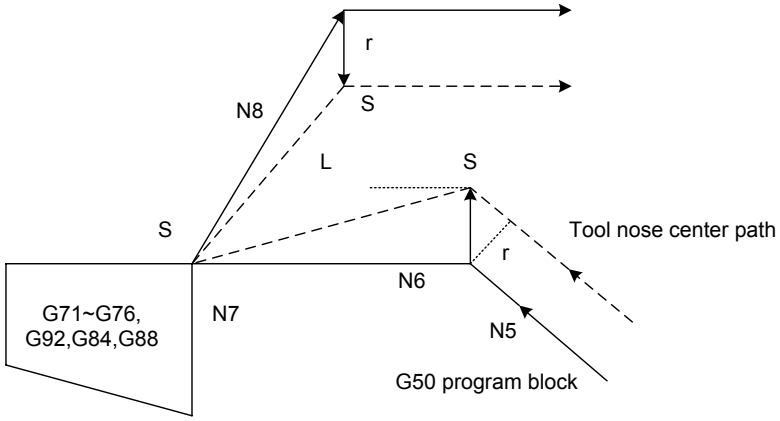


Fig. 4-60 Cancel compensation vector temporarily in cycle pause

- **G32, G34 thread cutting**

The system does not execute the tool nose radius compensation and temporarily cancels the tool nose radius compensation in G32, G34, and it automatically recovers the compensation mode in G00, G01.

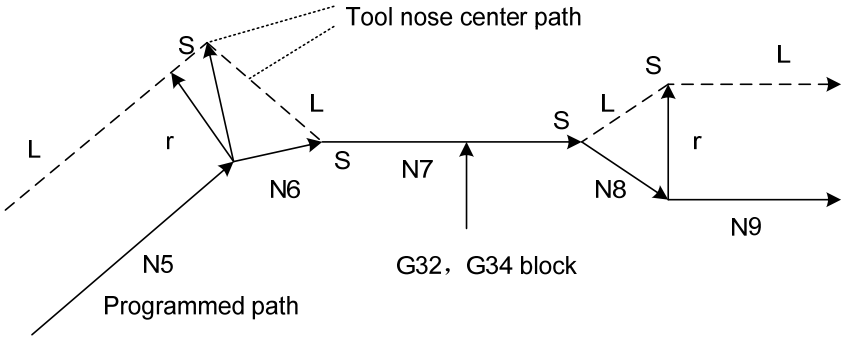


Fig.4-61 cancelling compensation vector in G32, G34 pause

- **G90, G94**

Compensation method of tool nose radius compensation in G90 or G94:

A. Each cycle path and tool nose center path are parallel to program path.

B. Offset directions are the same in G41 and G42 as the following figure, and the system determines the tool compensation direction according to the UW direction of starting point and end point, and executes the tool compensation according to the direction in the cycle process.

C. In having creating C tool compensation state, the system firstly cancels C tool compensation state in G90, G94, and executes the infeed tool to the intersection point of the tool nose center based on the tool nose center parallel programmed path, and at last to the positioning point. The system creates C tool compensation again in the next G00, G01.

D. After the system cancels the tool radius compensation, the imaginary tool nose point moves to the positioning point, and when the tool is in the cycle inner, the tool diameter exceeds the length of the rapid traverse of the first block, the overcut creates and No.255 alarms.

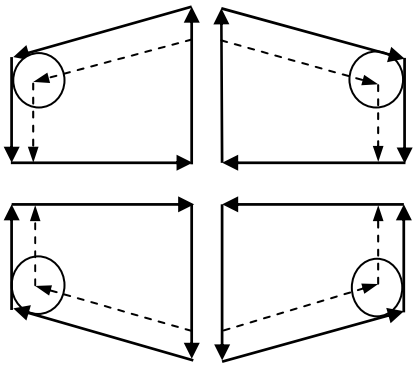


Fig. 4-62 Offset direction of tool nose radius compensation in G90

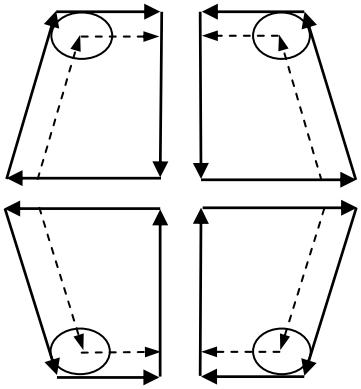


Fig. 4-63 Offset direction of tool nose radius compensation in G94

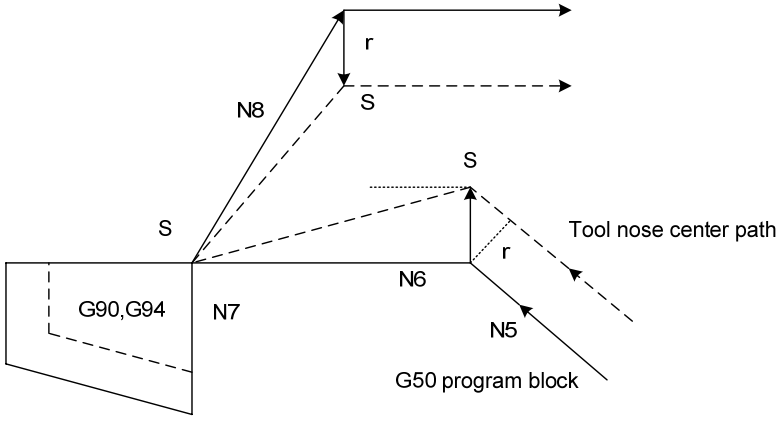


Fig. 4-64 G90, G94 radius compensation mode

- **G70 command**

When G71～G73 is executed, the system temporarily cancels C tool compensation. When G70 is specified again, the system automatically recovers the compensation mode. Because the system executes G71～G73, it does not execute the radius compensation, there must be the finishing allowance in programming to avoid the overcut in roughing.

In G70, the compensation mode is not cancelled after the cycle end, the system continuously executes the compensation in the fixed point, which causes the undercut of the finishing cycle in the last block, so, the last should exceeds one tool radius value of the workpiece in programming.

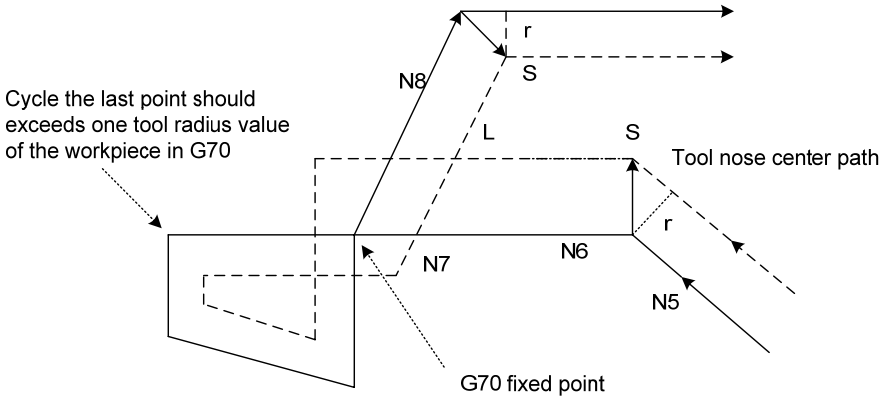


Fig. 4-65 G70 radius compensation mode

4.2.7 Particulars

- **Inside chamfer machining less than tool nose radius**

At the moment, the tool inside offset causes an excessive cutting. The tool stops and the system alarms (P/S41) when starting the previous block or chamfer moving. But the tool stops the end point of previous block when **Single** is ON.

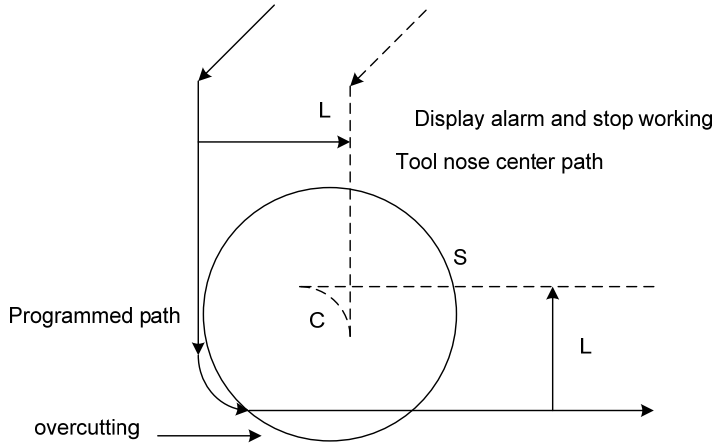


Fig.4-66 inner corner machining less than tool nose radius

- **Machining concave less than tool nose diameter**

There is an excessive cutting when the tool nose center path is opposite to program path caused by tool nose radius compensation. At the moment, the tool stops and the system alarms No.257 when starting the previous block or chamfer moving.

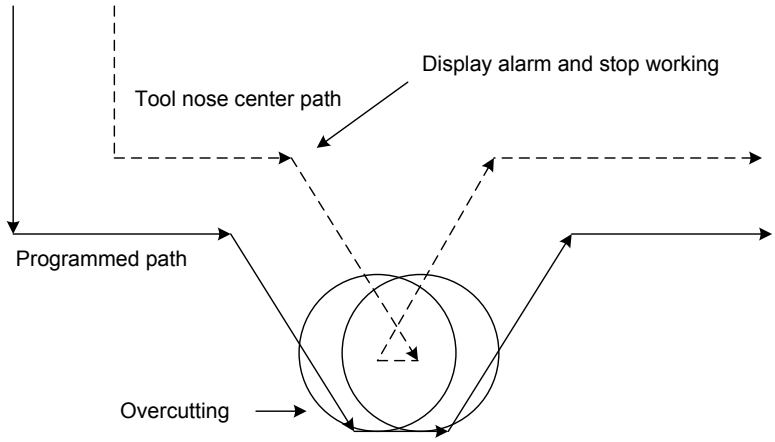


Fig. 4-67 machining a grooving less than tool nose radius

- **Machining a inner sidestep less than 90°**

When the system machines a inner sidestep less than or equal to 90° and the machining path length is less than the tool nose radius, there will be the too much undercut and No. 260 alarms. At the moment, No.5008 Bit6 (CNS) sets whether the system alarms in the condition.

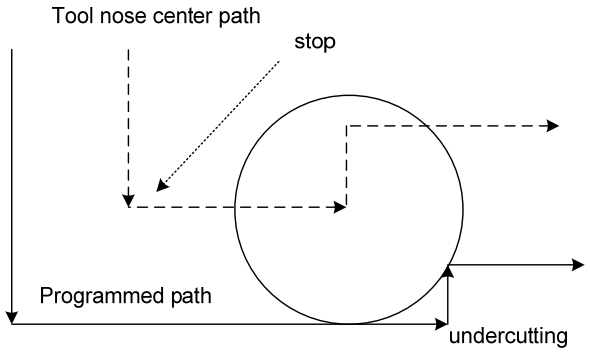


Fig.4-68 machining an inner sidestep less than 90°

- **Corner motion**

When two or more than movement vector in the end point of one block create, the tool moves to another vector from the vector linear, which is called the corner motion. When the single block is valid, the tool stops in the last vector.

When two vectors coincide, the system does not execute the corner motion and the second vector will be ignored. When the two-axis increments of the movement vector in the compensation level are less than the setting values of No. 5010(CLV), the second vector is ignored, but it is not ignored when the interpolation block is the arc.

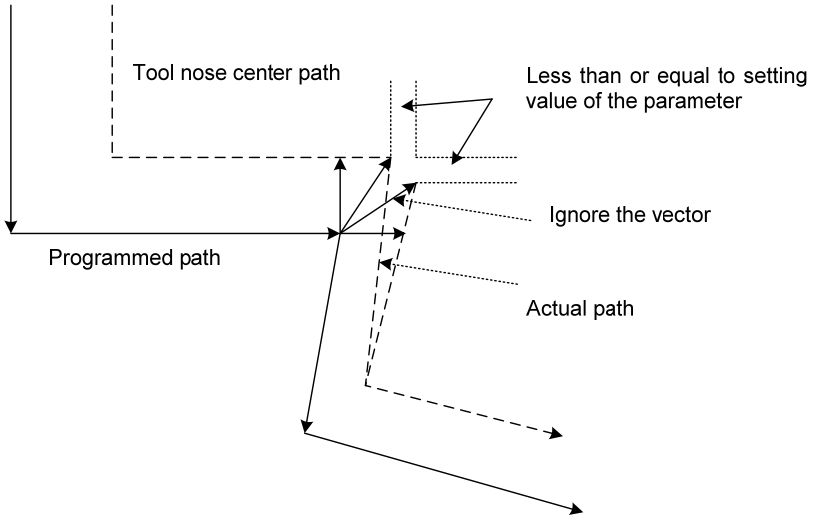

Tool nose center path
Less than or equal to setting value of the parameter
Ignore the vector
Programmed path
Actual path

Fig. 4-69 corner motion

- **Changing compensation value**

(a) The system executes the tool change in the compensation cancel mode, the compensation value is changed. When the compensation value is changed in the compensation mode, No.5001 Bit4(EVR) can set whether the compensation value change is valid from the nest T command or the next buffer block.

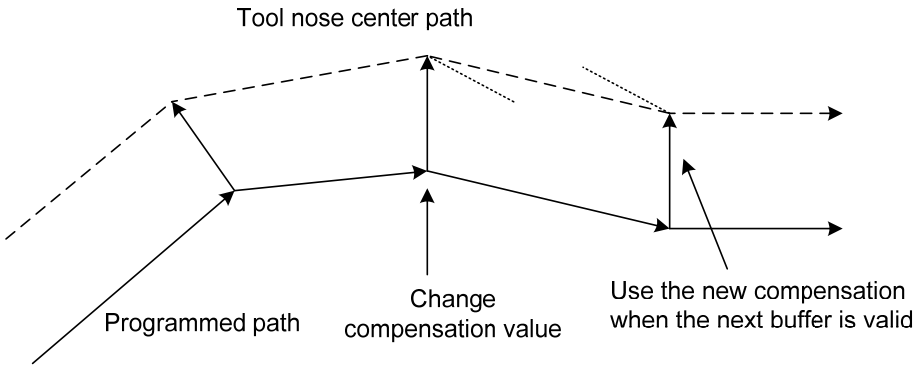


Fig.4-70 changing compensation value

(b) Positive/negative compensation value and tool nose center path

When the compensation value is negative (-), G41 and G42 exchange in programming. When the tool center moves along the workpiece outer, it moves along the inner, and vice versa.

Note: The compensation value is equal to the offset value adding the wear value. When the compensation sign is changed, the tool nose offset direction changes but the imaginary tool nose direction does not change. So, do not change the compensation sign optionally.

- **End point of programmed arc is not in the arc**

In the radius compensation process, when the system uses IJK to specify the circle center and the end point of the arc is not in the arc, the system positions again the circle center position specified by IJK, and confirms the circle center position according to the radius counted by IJK to execute the radius compensation. When the counted radius is too small not to reach the end point of the arc, No. 254 alarm creates.

Note: At the moment, there is a difference between the counted arc and the specified in programming, and the function is sued to regulate the error of the radius out-of-tolerance in some range in programming.

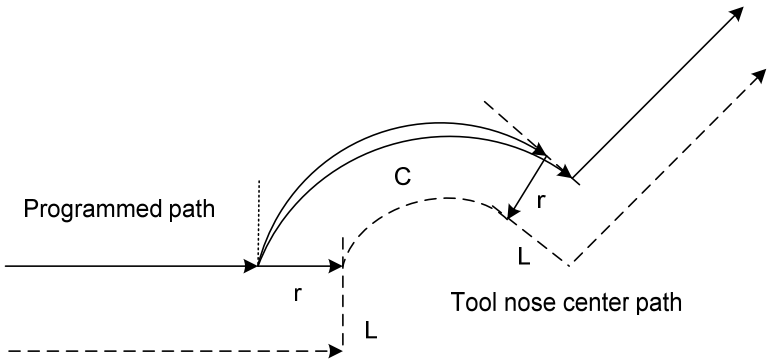


Fig. 4-71 End point of programmed arc be not in the arc

- **Continuous 3 or more than 3 blocks non-movement command**

In tool radius compensation process, when there are 3 or 3 blocks without movement command, the tool nose center reaches the end point of the last block and is vertical with programmed path position of the last, at the moment, which causes the overcut, and the programmer should pay more attention it.

General non-movement command ：

1. M03S300 only have M, S, T, F, O, N codes
2. #100=3 non-NC statement （when 6000#5 SBM is set to 1）

3. G04 X10 pause
4. G00 only have G code and do not specify the position command
5. G01 U0 the infeed distance is 0
6. G01X100 only specify the absolute value which is same with that of the last block
7. G01Y10 only specify the axis in non tool compensation level
8. M98M99 statement for calling subprogram and subprogram return(the block has no axis increment command）
9. G66G67 statement for calling macro program and cancelling macro program call modal
10. ; Null block

In non-movement block, when there is a command to cancel the radius compensation, the system does not cancel the vector and execute the command in the vertical vector. It cancels the radius compensation vector when the system cancels the radius compensation in G28, G30, G53, it executes the command in the vertical vector in G50, G52, G32, G34, fixed cycle, multi cycle, drilling cycle and other commands.

When there are 3 or more than 3 blocks without movement command following the block used to create the tool radius compensation, the system does not immediately create the tool radius compensation but does it in the block following the non-movement command.

The system executes the above vertical before the last movement command when there is a optional symbol "/" in tool radius compensation. So, please do not use the optional block function in the tool radius compensation to avoid the overcut.

When No.6000 Bit5 (SBM) is set to 1, the macro statement can stop in single block and is taken as the non-movement block in the tool nose radius compensation at the moment, which causes the abnormal path. It is suggested that No.6000 Bit5 (SBM) is set to 0 when the system uses the macro statement in the tool nose radius compensation mode in the course of normal machining.

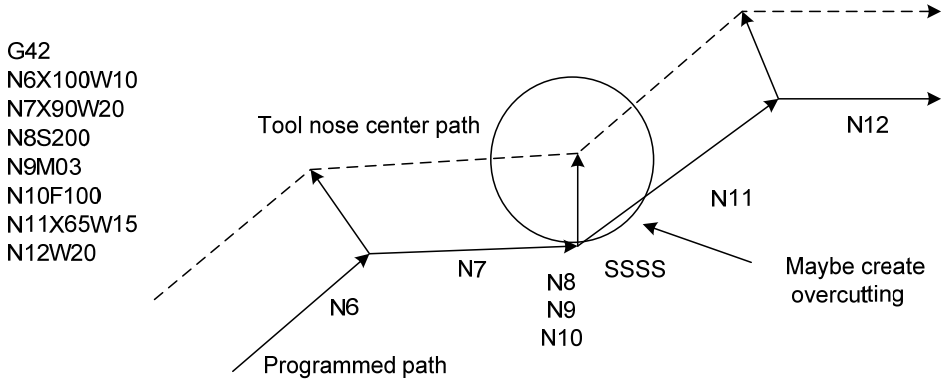


Fig. 4-72 continuous 3 or more than 3 blocks of non-movement command

- **Subprogram call and macro statement call in G code**

In tool nose radius compensation mode, when the system specifies the code for calling the subprogram, it can execute the normal compensation, the compensation method of calling program is transferred to the subprogram which is to execute the corresponding compensation.

The command for calling subprogram and subprogram return has no movement command, it is taken as the non-movement block. When the system specifies G code in the radius compensation mode in the subprogram, G code is valid, at the same time, the system cancels the radius compensation mode when the subprogram does not end, the compensation mode is transferred to the called program which will continuously executes the corresponding compensation.

- **Cutting inner of the whole circle**

In the tool nose radius compensation, when the system machines the inner of the whole circle and the compensation direction is not changed, the overcut or undercut creates, at the moment, it determines whether it alarms based on No.5008 Bit5 (CNF). When Bit5 is set to 0, No.259 alarms.

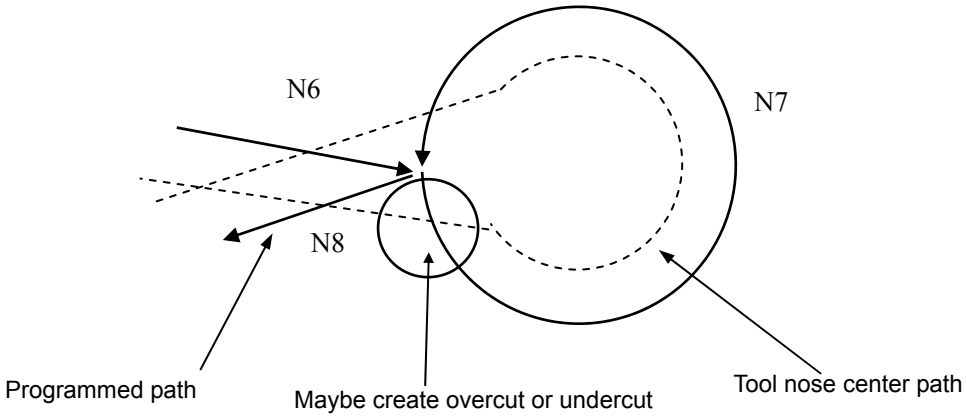


Fig. 4-73 overcut in machining inner of the whole circle

- **Inserting MDI operation in tool compensation**

In MDI mode, the system does not execute the tool nose radius compensation. When the system specifies G41 or G42, the system determines No.5008 Bit4 (MCR). When Bit is set to 1, No.258 alarms. The system does not alarm and ignores the specified G41 and G42 when it is set to 0,

When the system runs in AUTO mode in absolute command programming and the single block run stops to insert MDI mode, and then starts AUTO mode, at the moment, transfers the vector of starting point of the next block, and forms other vectors based on the next two blocks, the offset can be executed from PC, and the tool path is as follows:

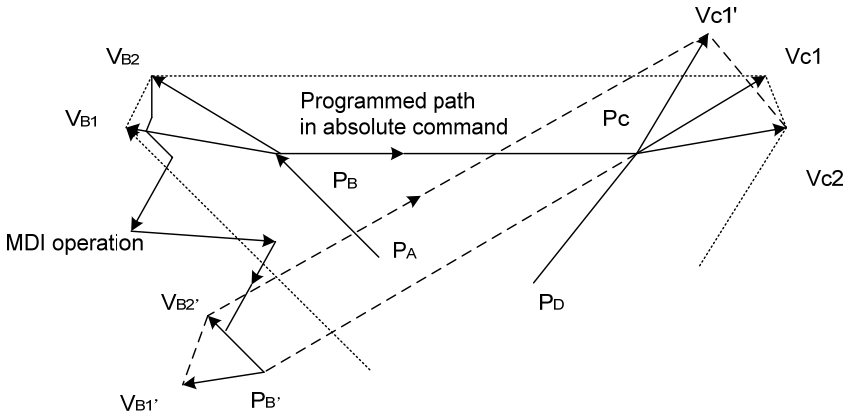


Fig. 4-74 insert tool offset of block in MDI mode

When PA, PB, PC is programmed with absolute command, the single block run stops and the tool is moved in MDI after the block from PA to PB is executed. The vector VB1, VB2 are transferred to VB1' and VB2', VC1' ,VC2 of PB'→PC and PC→PD are calculated again.

But, the system can correctly execute the compensation following PC because the vector VB2 has not calculated again.

Ⅱ OPERATION

Chapter Ⅰ Overview

1.1 Operation Overview

GSK988T has operation modes including EDIT, AUTO, MDI, REFERENCE POSITION RETURN, MPG/STEP, MANUAL, DNC and so on.

- **Editing a program**

The above-mentioned operation is completed by the program edit function. The edit program is saved to the memory of the system, and the program can be modified and altered.(see Chapter Ⅴ).

- **Automatic run**

The automatic run is to operate the machine based on the compiled program. Once the program is compiled to the CNC memory, it runs according to the program command. This operation is called the automatic run. (See Chapter 6.1) program.

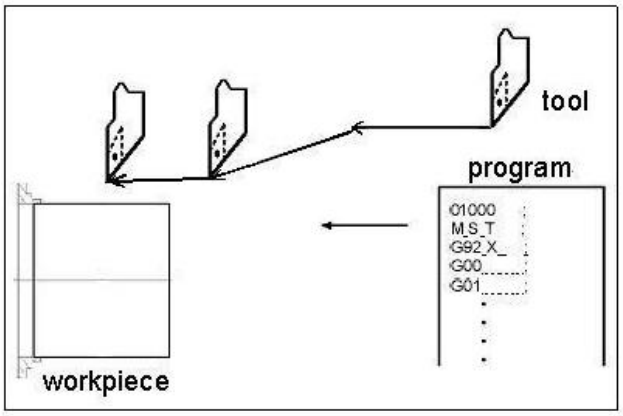


Fig.1-1 Automatic run

- **MDI run**: After a program is input in MDI window, the machine runs according to the program command, and this operation is called MDI mode run. (See Chapter 6.2).

- **Reference position return**

CNC machine has a special point which is used to determine the position of the worktable of the machine. The point is called the reference position, at which the tool change is executed or the coordinate system is set. After the power is turned on, the tool traverses to the reference position. The manual reference position return is to traverse the tool to the reference position by the switch and the button on the operation panel. (See 4.1).

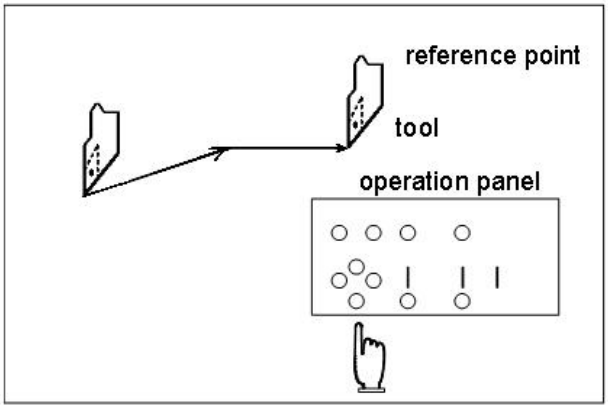


Fig.1-2 Manual reference position return

The tool traverses to the reference position by the program command, which mode is called the automatic reference position return.(See Programming).

- **MPG feed**

The tool traverses an distance which corresponds to the rotary angle by rotating the MPG. (See 5.4).

- **Manual run**

The tool runs along each axis by the switch, the button or the MPG on the machine operation panel.

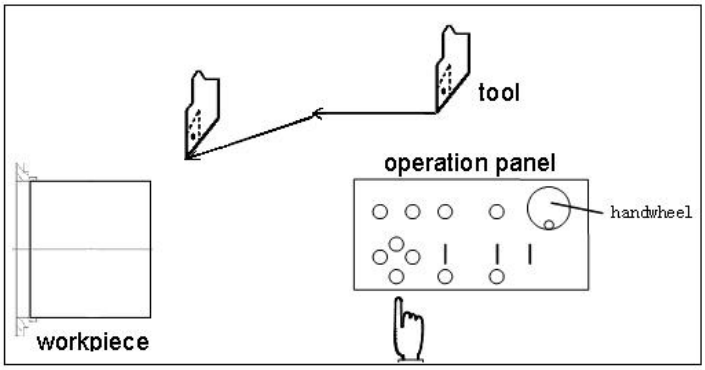


Fig.1-3

（1）Manual（manual continuous）feed（see 5.2）

When the press key is kept down, the tool continuously traverses.

（2）Incremental feed（see 5.3）

The tool only traverses some distance when the key is pressed once.

- **DNC run:** The system directly reads the programs to run the machine by the external input/output device instead of that the program is saved to the CNC memory. (See 6.3)

1.2 System Setting

The operator executes a series of setting to the CNC by its press keys, and the common setting including: tool offset, CNC setting and macro variable setting.

- **Tool offset setting:** each tool has its own dimension (length, diameter) . When a workpiece with some shape is machined, the tool dimension is different according to the amount of movement. If the dimension Value of the tool is set in the CNC, even if the different tools are used, the tool path is also automatically given in the same program, so any tools can machine the workpiece shape specified by the programs.

The Value related to the tool dimension is called the offset. (See Chapter Ⅷ)

- **CNC setting:** CNC setting includes: system setting, coordinate setting, system time setting, system IP setting. (See Chapter 3.4)
- **Macro variable setting:** The system can support all kinds of macro program edit, the variable required by the macro program is set here.

1.3 Display

➢ **Program display:**

1. Display the current program content being executed shown in Fig.1-4. clue to function

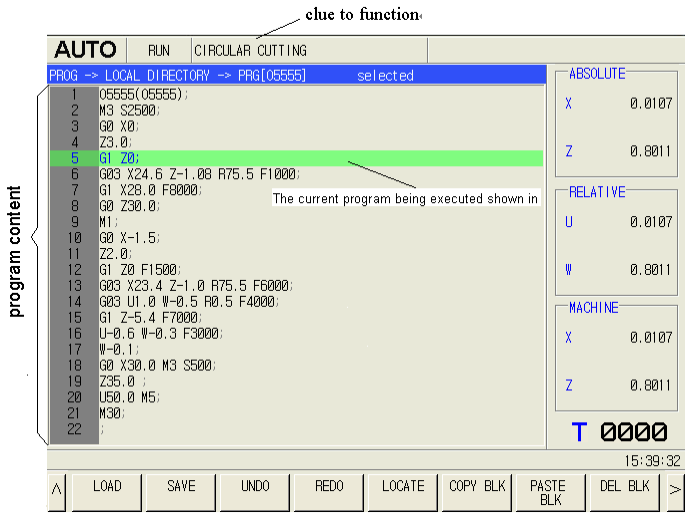


Fig.1-4

2. CNC stores programs shown in Fig.1-5.

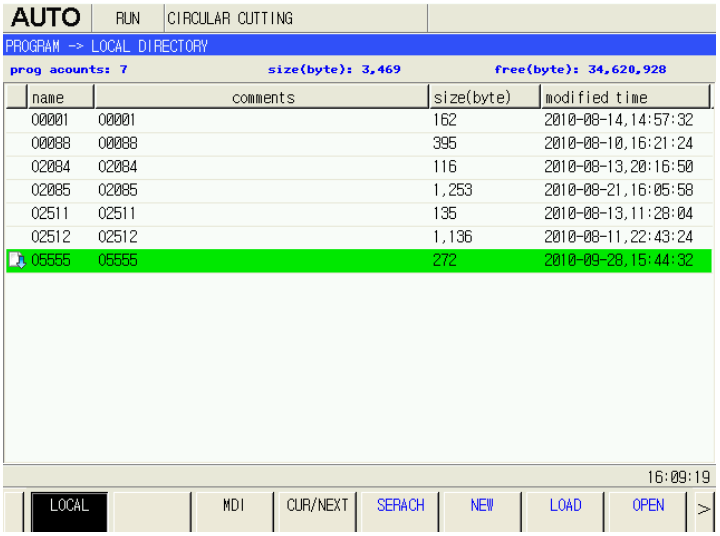


Fig.1-5

➢ **Current coordinate display**

The coordinate values of each coordinate system display the position where the tool is, and can be taken as the distance display from the current to the target position shown in Fig. 1-6. (See Chapter 3.1 Position Display Window).

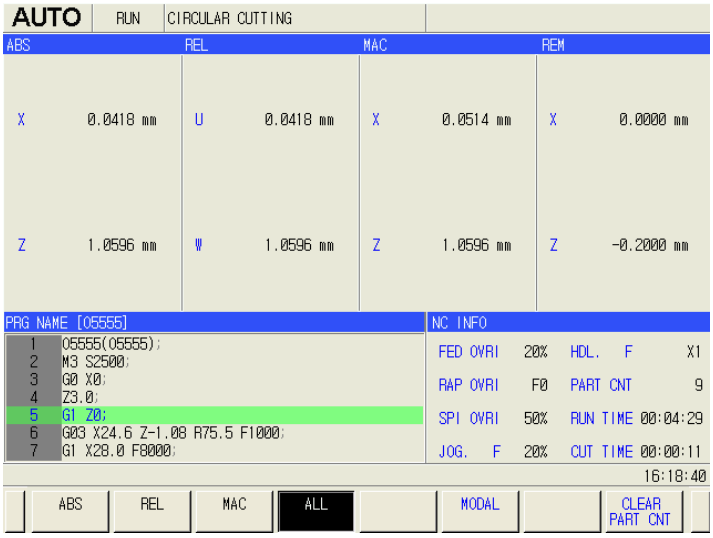


Fig.1-6

➢ **Displaying alarm**

When the failure occurs in the course of run, the corresponding mistaken commands and the alarm message are displayed in the window shown in Fig.1-7. The detailed explanations related to the alarm message are referred to Appendix Ⅰ.

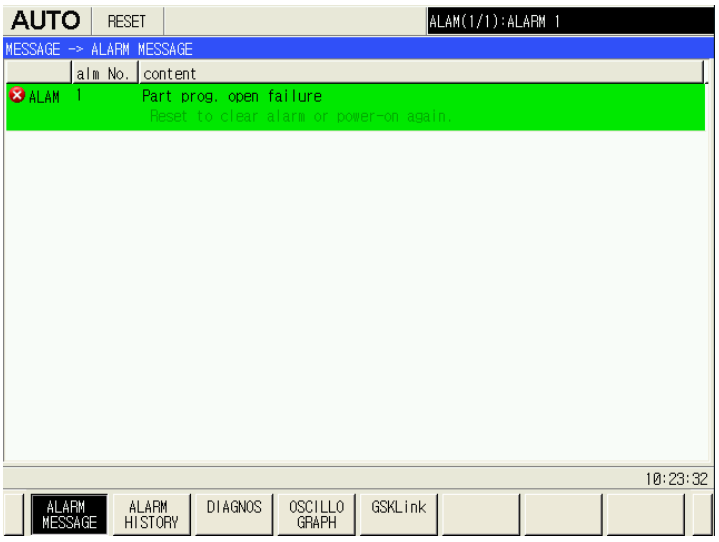


Fig.1-7

➢ **Displaying machined workpiece count and operation time**

Display the machined workpiece count, run time and cutting time in the current position display window shown in Fig.1-8: out time

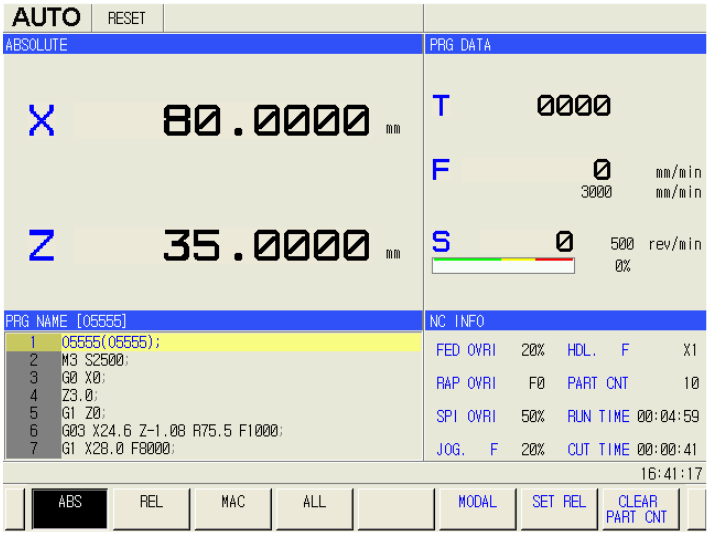


Fig.1-8

1.4 System

1.4.1 System panel

GSK988T system panel adopts 8.4″LCD and its appearance is shown in Fig.1-9:

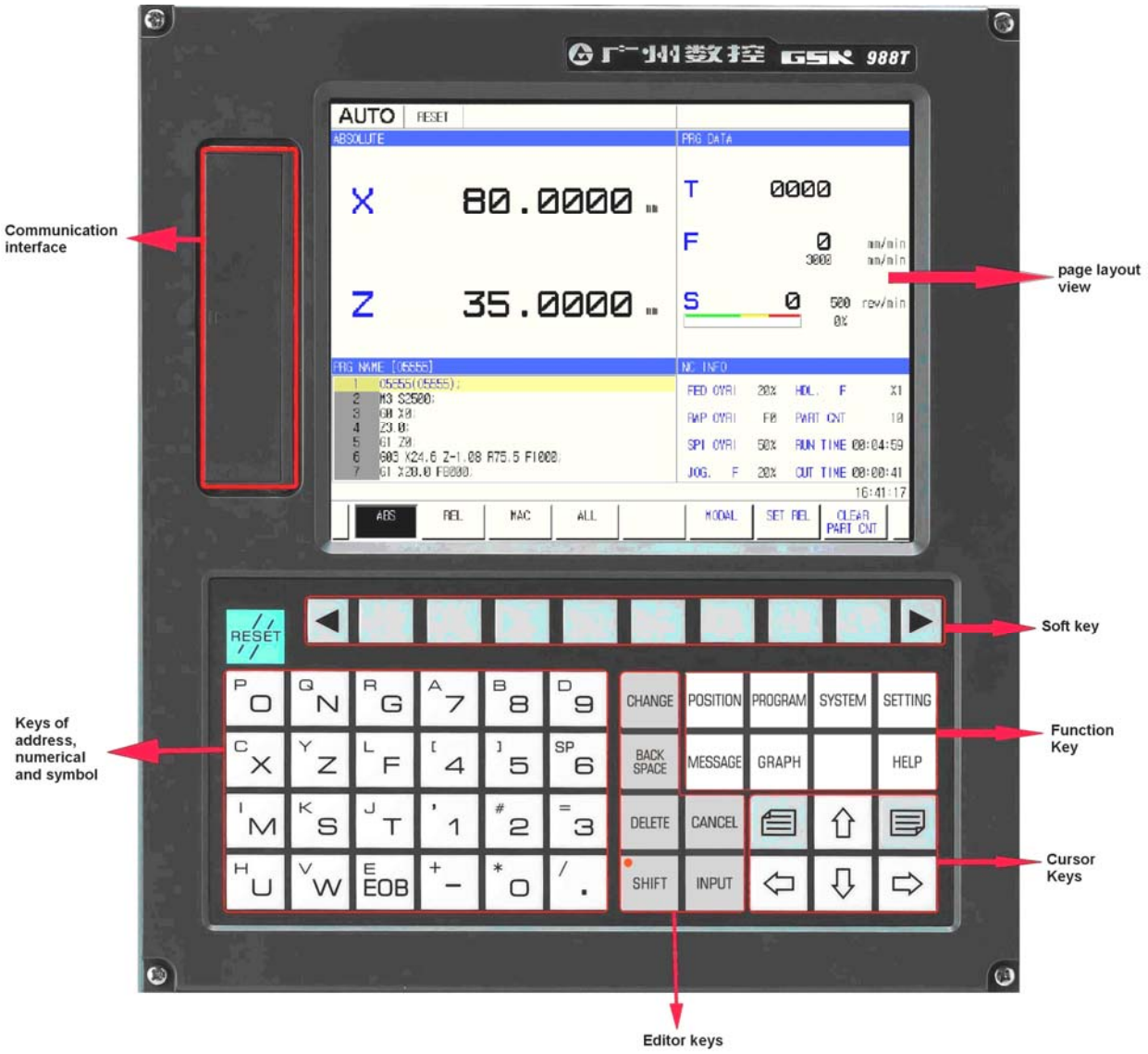
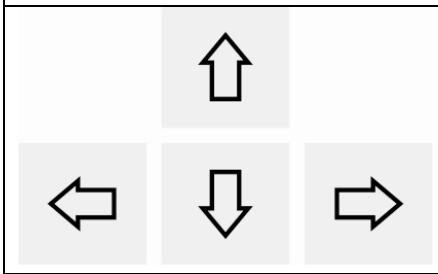
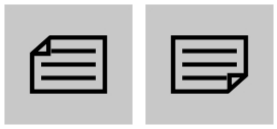
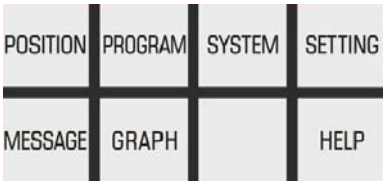
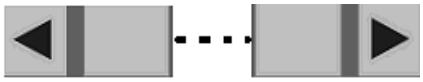
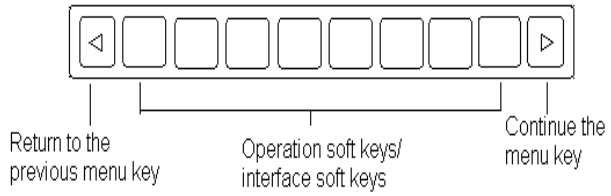


Fig. 1-9

1.4.2 System key definitions

Keys	Name	Introduction of the function
RESET	Reset key	CNC reset, feeding and output stop, etc
P O Q N R G A 7 B 8 D 9 C X Y Z L F [4] 5 SP 6 I M K S J T ' 1 # 2 = 3 H U V W E EOB + − * 0 / .	Keys of address, numerical and symbol	Input the address, number and symbol, press shift key and take the above address or symbol; Otherwise, take the address below.

Keys	Name	Introduction of the function
SHIFT	Shift key	Switch among keys of double addresses, double symbols, address symbol and numerical address, firstly press shift key and its indicator is on, and then press address key, input the address above; or select one block with the cursor keys
INPUT	Input key	Input the Value of parameter and compensation value, etc, and switch the line during editing the program.
CHANGE	Change key	Switch between the message and the display, with function of Tab key, and forming the shortcut keys with the other keys during editing the program.
BACK SPACE	Backspace key	Delete the program and the character, etc ahead
CANCEL	Cancel key	Cancel the operation
DELETE	Delete key	Cancel the program and the character, etc backward
	Cursor keys	Control the cursors to move up, down, left and right
	Window key	Switch the windows in one window
	Function key	POSITION Press it to switch the position display window. PROGRAM Press it to switch the program display window. SYSTEM Press it to switch the system display window.

Keys	Name	Introduction of the function
		Press it to switch the system display window. Press it to switch the message display window. Press it to switch the graph display window. Custom window . Press it to switch the help display window.
	Soft key	After using the function keys switches the windows, using the corresponding soft key can display the content of some sub-page in the current window or some operations in the current window are executed. The soft keys of GSK988T have 10 in the below of the screen shown below.  Soft key function: ① Switch the sub-pages in the current window; ②Operate on the current secondary window, such as editing and rewriting the Value or displaying the content, etc.

1.5 Machine Operation Panel

1.5.1 Division of machine operation panel

GSK988T matches two kinds of operation panel including MPU02A and MPU02B, MPU02A is with MPG and MPU02B is without MPG shown in Fig.1-10:

Fig.1-10

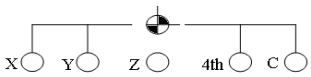
Note: The operations related to the machine operation panel described in the manual has two types, and when the panel allocated by the user is different from the two, please refer to the attached message.

1.5.2 State indicator and press key definition on the panel

The function of keys on GSK988T machine panel is defined by PLC program (ladder diagram), and about the detailed function of each key, refer to the manual of the machine manufacturer. The machine panel is taken as the reference.

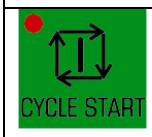
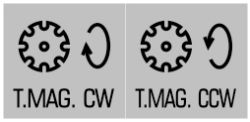
Function of the machine panel each key defined by GSK988T standard PLC program, refer to the following list:

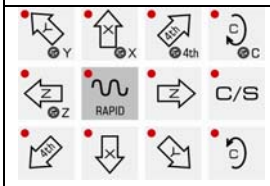
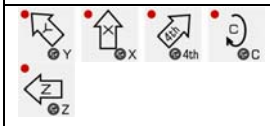
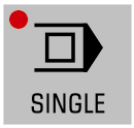
State indication

	The indicator for each axis reference position return	RUN	Running indicator
ALM	Alarm indicator	L1~L5	Self-defined indicator
GEAR/TOOL NO.	Gear/tool number indicator		

Press key definition:

Keys	Names	Function	Mode during valid function

Keys	Names	Function	Mode during valid function
	Feed hold key	The program and MDI command running pause	Auto mode, MDI mode and DNC mode
	Cycle start key	The program and MDI command running start	Auto mode, MDI mode and DNC mode
	Feedrate override knob	Adjusting the feedrate	Auto mode, MDI mode, edit mode, reference position return mode, MPG mode, single step mode, manual mode and DNC mode
   	Spindle override keys	Adjusting the spindle speed only when the spindle speed analog value control mode is valid	Auto mode, MDI mode, edit mode, reference position return mode, MPG mode, single step mode, manual mode and DNC mode
	Feedrate override knob	Adjust the feedrate	Auto mode, MDI mode, Edit mode, Reference position return mode, MPG mode, Step mode, DNC mode.
	Manual tool change keys	Manual tool change	Reference position return mode, MPG mode, single step mode and manual mode
	Position record	Record current coordinate position used to input the tool offset	Auto mode, MDI mode, Edit mode, Reference position return mode, MPG mode, Step mode, DNC mode.
	Jog key	Spindle jog on/off	MPG mode, single step mode and manual mode
	Lubricating key	Machine lubricating on/off	Auto mode, MDI mode, edit mode, reference position return mode,

Keys	Names	Function	Mode during valid function
	Cooling key	Cooling on/off	MPG mode, single step mode, manual mode and DNC mode
	Chuck key	Chuck clamp/release	Auto mode, MDI mode, edit mode, reference position return mode, MPG mode, single step mode, manual mode and DNC mode
	Spindle keys	Spindle rotation CCW Spindle stop Spindle rotation CW	MPG mode, single step mode and manual mode
	Rapid speed switch	Switch between rapid speed/ feedrate	Auto mode, MDI mode, manual mode and DNC mode
	Manual feeding keys	each axis moving positive/negative in manual or single step mode	Reference position return mode, single step mode and manual mode
	MPG control axes option keys	Each axis option in MPG mode	MPG mode
	Option keys of MPG/single step increment and rapid override	MPG movement value of each grid: 0.001/0.01/0.1/1 mm Single step movement value of each step: 0.001/0.01/0.1/1 mm Rapid override: F0, F25%, 50% and F100%	Auto mode, MDI mode, reference position return mode, MPG mode, single step mode, manual mode and DNC mode
	Single block switch	Switch between the single block running/continuous running, when the single block is valid, its indicator is on.	Auto mode, MDI mode and DNC mode
	Block skip switch	Whether skip and switch the block with"/" at the beginning; When the block skip switch is on, its indicator is on.	Auto mode, MDI mode and DNC mode
	Machine lock switch	When the machine is locked, its indicator is on and each axis output is invalid	Auto mode, MDI mode, edit mode, reference position return mode, MPG mode, single step

Keys	Names	Function	Mode during valid function
			mode, manual mode and DNC mode
	Miscellaneous function lock switch	When miscellaneous function is locked, its indicator is on and the function of M, S and T output is invalid	Auto mode, MDI mode and DNC mode
	Dry run switch	When dry run is valid, its indicator is on and the machine program/MDI command block begins dry running	Auto mode, MDI mode and DNC mode
	Optional stop key	When optional stop is valid, its indicator is on; when there is M01 in the block, move to the block and the running stops	Auto mode, MDI mode and DNC mode
	Edit key	Access edit mode	Edit mode
	Auto key	Access the auto mode	Auto mode
	MDI key	Access MDI mode	MDI mode
	Reference position return key	Access reference position return mode	Reference position return mode
	Single step/MPG key	Access single step or MPG mode (One mode is selected by parameter)	Single step mode/MPG mode/manual mode
	Manual key	Access the manual mode	Manual mode
	DNC key	Access DNC mode	DNC mode
	Feed/spindle hold knob	Feed/axis hold function	Auto mode, MDI mode, edit mode, reference position return mode, MPG mode, single step mode, manual mode, DNC mode

Keys	Names	Function	Mode during valid function
	Emergency stop key	In emergency, the system and the machine stop running, all output is closed.	Auto mode, MDI mode, edit mode, reference position return mode, MPG mode, single step mode and DNC mode
 	Power on/off keys	System power of/off switch	Auto mode, MDI mode, edit mode, reference position return mode, MPG mode, single step mode, manual mode and DNC mode
	Overtravel release key	Cancel machine limit	MPG mode and manual mode
	Program protection switch	The protection program can’t be changed at random.	Auto mode, MDI mode, edit mode, reference position return mode, MPG mode, single step mode, manual mode and DNC mode
	MPG key	Control the machine movement	MPG mode

Chapter Ⅱ Power on, Power off and Safety Protection

2.1 Power on

Before GSK988T powers on, they should be confirmed:

1. The machine is normal;
2. The power supply and the voltage comply with the requirements;
3. The connection is right and fixed.

After GSK988T powers on, the window is shown as below:

Fig.2-1

Then, GSK988T self-detects and initializes. After the system completes the self-detection and the initialization, the window of the present position (absolute coordinate) displays.

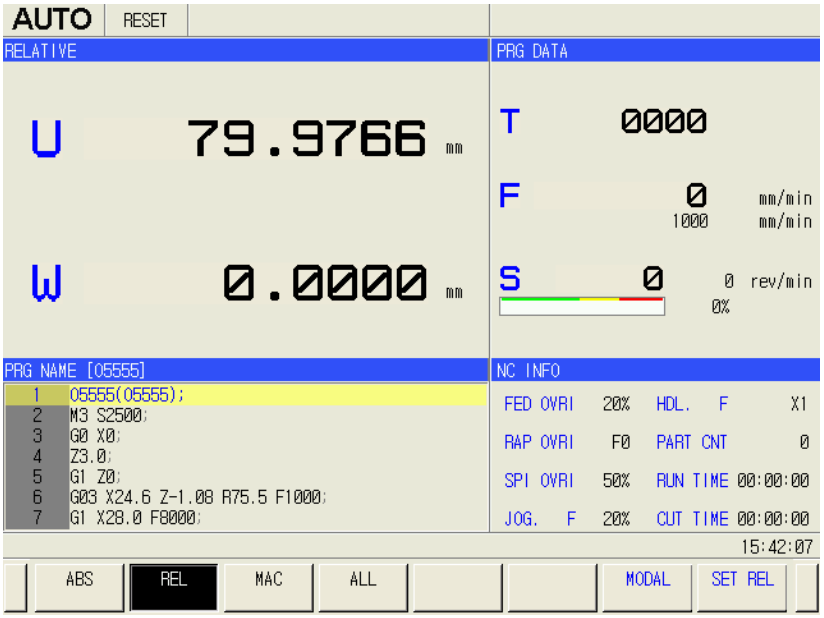


Fig. 2-2

2.2 Power off

Before power off, they should be confirmed:

1. Each axis of CNC stops;
2. The miscellaneous function switches off (such as the spindle and the water pump, etc)
3. Firstly cut off CNC power supply, and then cut off machine power supply.

Note 1 : The system can be restarted again after the power is hold OFF for 20m.
Note 2: About the operation of cutting off the machine power supply, refer to the manual of the machine manufacturer.

2.3 Overtravel Protection

To avoid the damage of the machine due to the overtravel for each axis, the machine must take the measure of overtravel protection.

Install the limit switches on the maximum stroke in each axis positive and negative directions on the machine. When it overtravels, the limit switch is on, the system decelerates till stopping and it alarms overtravel.

During auto running, when the machine moves along one axis and touches the limit switch, the tool decelerates and stops as long as it traverses along all axes and the system alarms overtravel.

During the manual operation, only the axis which the tool touches its limit switch decelerates and stops, while the tool still traverses along other axes.

The method of canceling the alarm of “overtravel”: In the manual mode, the working table moves in the opposite direction (For example: Overtravel is in the position direction, it moves negatively; negative, positively.) and leaves off the limit switch. Reset, the alarm is cleared.

Note: The overtravel release method on the machine is different that of the User Manual, and the concrete operations are referred to the machine manufacturer’s.

2.4 Overtravel Protection in Memory Travel Limit

The tool can’t enter the area stipulated by the travel limit check 1, 2 and 3 in memory type.

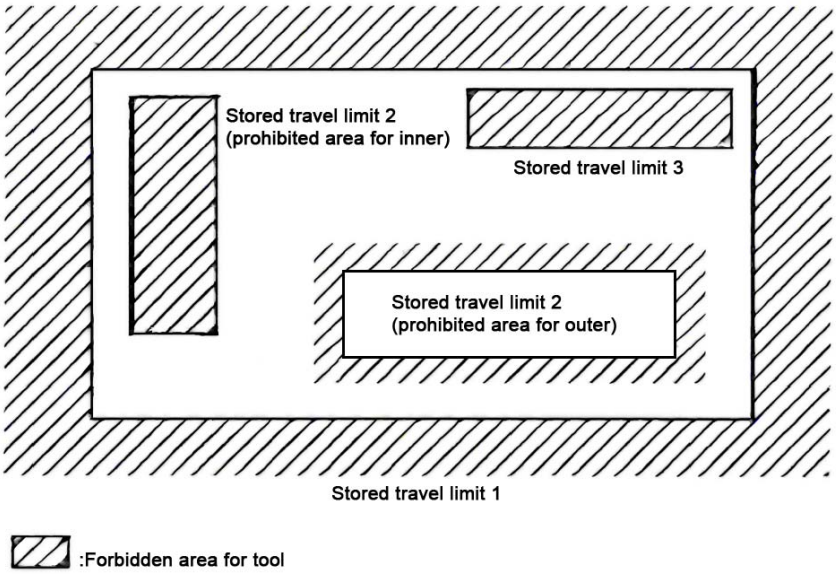

Stored travel limit 2
(prohibited area for inner)
Stored travel limit 3
Stored travel limit 2
(prohibited area for outer)
Stored travel limit 1
:Forbidden area for tool

Fig.2-3

When the tool exceeds the travel limit in memory type, it alarms and the tool decelerates and stops. When the tool enters the forbidden area and alarms, the tool can traverse in the opposite direction.

Travel limit check 1 in memory type: The board is set by parameter (#1320 and #1321 or #1326 and #1327); the outside of the range is set as the forbidden area. The machine manufacturer normally sets the area as the maximum stroke.

Travel limit check 2 (G22 G23) in memory type: It is set by parameter (#1322 and #1323) or commands. During programming, G22 forbids the tool enters the forbidden area; G23 allows the tool enters the forbidden area. In the program, G22 and G23 should be specified independently, which are independent blocks; about the details, refer to the introduction of G commands.

Travel limit check 3 in memory type: The internal board of the area set by parameters #1324 and #1325 as the forbidden area.

Overlap of the forbidden area: Each forbidden area can be overlapped (refer to the following figure), but the outside of the machine travel isn't limited.

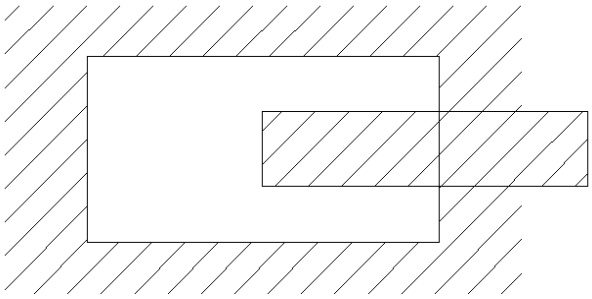


Fig.2-4

The valid time of the forbidden area: After connecting the power supply and manual reference position return or auto reference position return through G28, each limit becomes valid. After connecting the power supply, if the reference position is in the limited area, it alarms immediately.

Display the alarm time: It alarms immediately before or after the tool enters the forbidden area, which is set by the 7^{th} bit of #1300 of parameter BFA.

Overtravel alarm release: When the tool can't traverse in the forbidden area, switch into the manual mode and the tool traverses out of the forbidden area in the opposite direction (for example, overtravel is in the positive direction, it traverses negatively; negative, positively), press the resetting key, the alarm is cleared. If the setting is wrong, after rewriting and setting, the tool returns to the reference position.

Note: During setting the forbidden area, if two points are set as same, the area is as below:

1. **When the forbidden area is travel check 1 in memory type, all the areas are taken as the forbidden one.**
2. **When the forbidden area is travel check 2 or 3 in memory type, all the areas are taken as the movable area.**
3. **1300.7=1 and an alarm occurs, the machine coordinates are beyond the prohibited area, at the memoent, pressing RESET key can cancel alarm.**
4. **1300.7=1 and an alarm occurs, the machine coordinates are in the prohibited area, at the memoent, pressing RESET key can cancel alarm.**

When the stored travel limit 1 check is set and set values of coordinates in the positive is less than those the negative, the soft limit function disables.

2.5 Emergence Operation

During the processing, due to the user programming, operation and the product default, etc, some unexpected situations may occur, then, GSK988T should stop working immediately. In this chapter, it mainly introduces the measures taken in emergency. About the machine in emergency, refer to the relative introduction of the machine manufacturer.

2.5.1 Reset

When GSK988T output and the coordinate axis moves abnormally, press

and GSK988T resets:

1. All axes movement stops;
2. Function of M and S output invalid;
3. Auto running completes, the mode function holds.

Note: The parameter sets whether the system automatically closes the spindle CW/CCW, the lubricating, the cooling signal after

is pressed.

2.5.2 Emergency stop

During the machine running, in the dangerous or the emergency situation, press the emergency stop button and the external emergency stop signal is valid, and then CNC works in the emergency situation and the machine stops moving at once, all output is off, such as the revolving of the spindle and the cooling fluid. After releasing the emergency stop button, the alarm is released, CNC resets.

Note 1: Before releasing the emergency stop alarm, confirm the trouble is shot;
Note 2: Before power on and off, press the emergence stop button to reduce the electric shock of the equipment;
Note 3: After releasing the emergence stop alarm, return to the reference position again to ensure the precision of the coordinate position.

2.5.3 Feed hold

During the machine running, press

to stop the running, temporarily. Pay attention to that during the thread cutting or the cycle command running, even press the button, the running can't stop immediately.

2.5.4 Cutting off power supply

During the machine running in the dangerous situation or emergency, the machine power supply can be cut immediately to avoid the accidence. But, pay attention to that the coordinate displayed by CNC can't comply with its actual position after cutting off power supply, so it requires returning to the reference position, again.

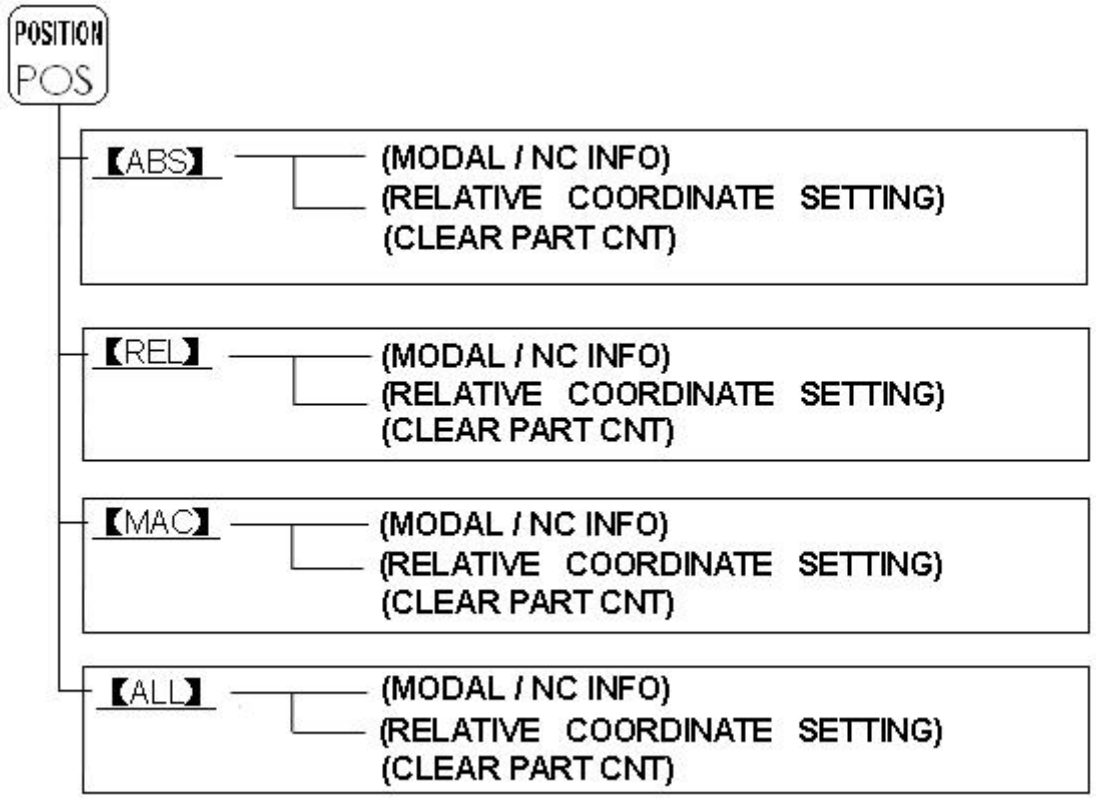
Chapter Ⅲ Windows

Based on the windows, this chapter introduces the relation among the switching windows, input and soft keys and the detailed operation method.

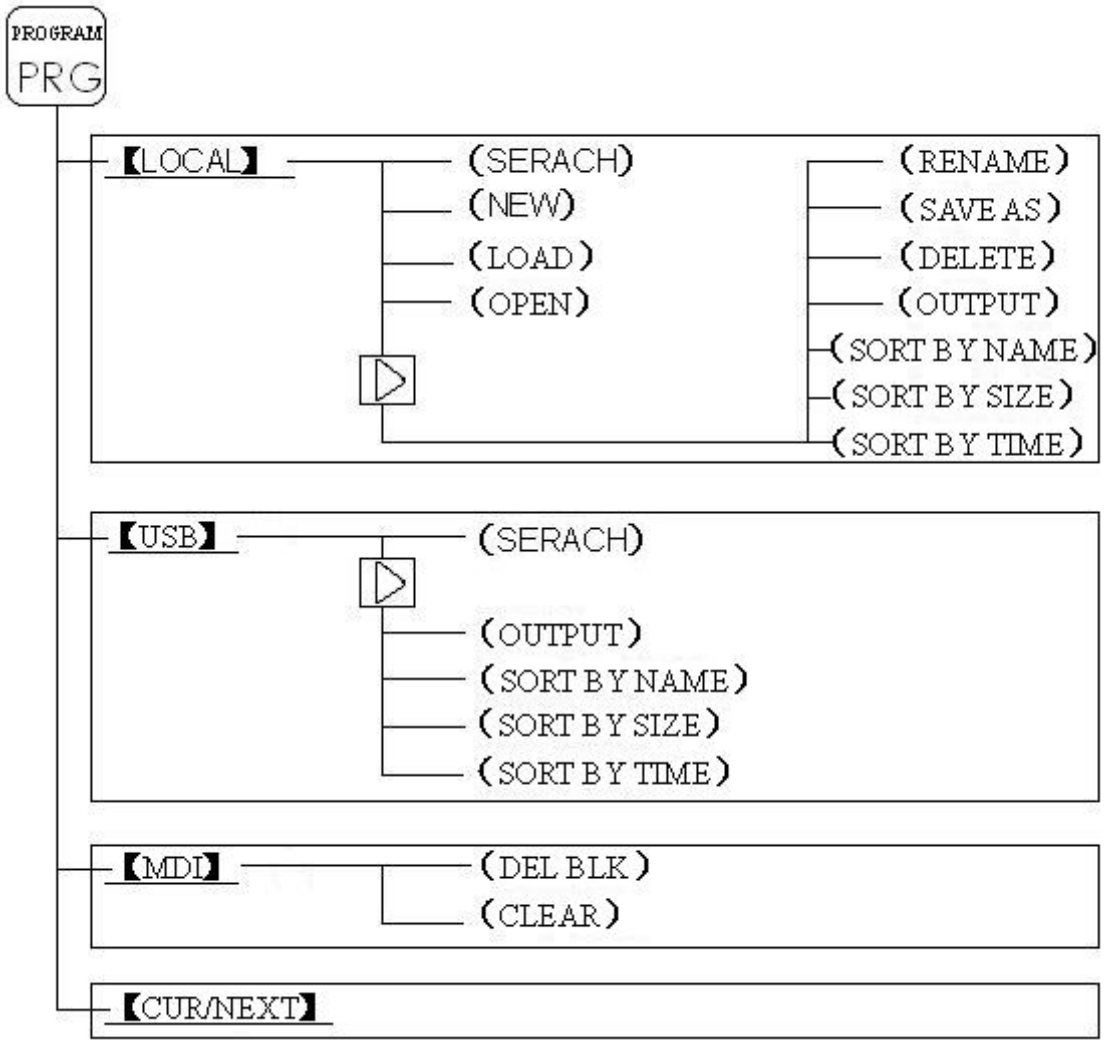
There are eight function keys including position, program and setting, etc on MDI panel in GSK988T system, each function key is relative to one main window, and each main window also includes many windows and the soft keys.

Note:

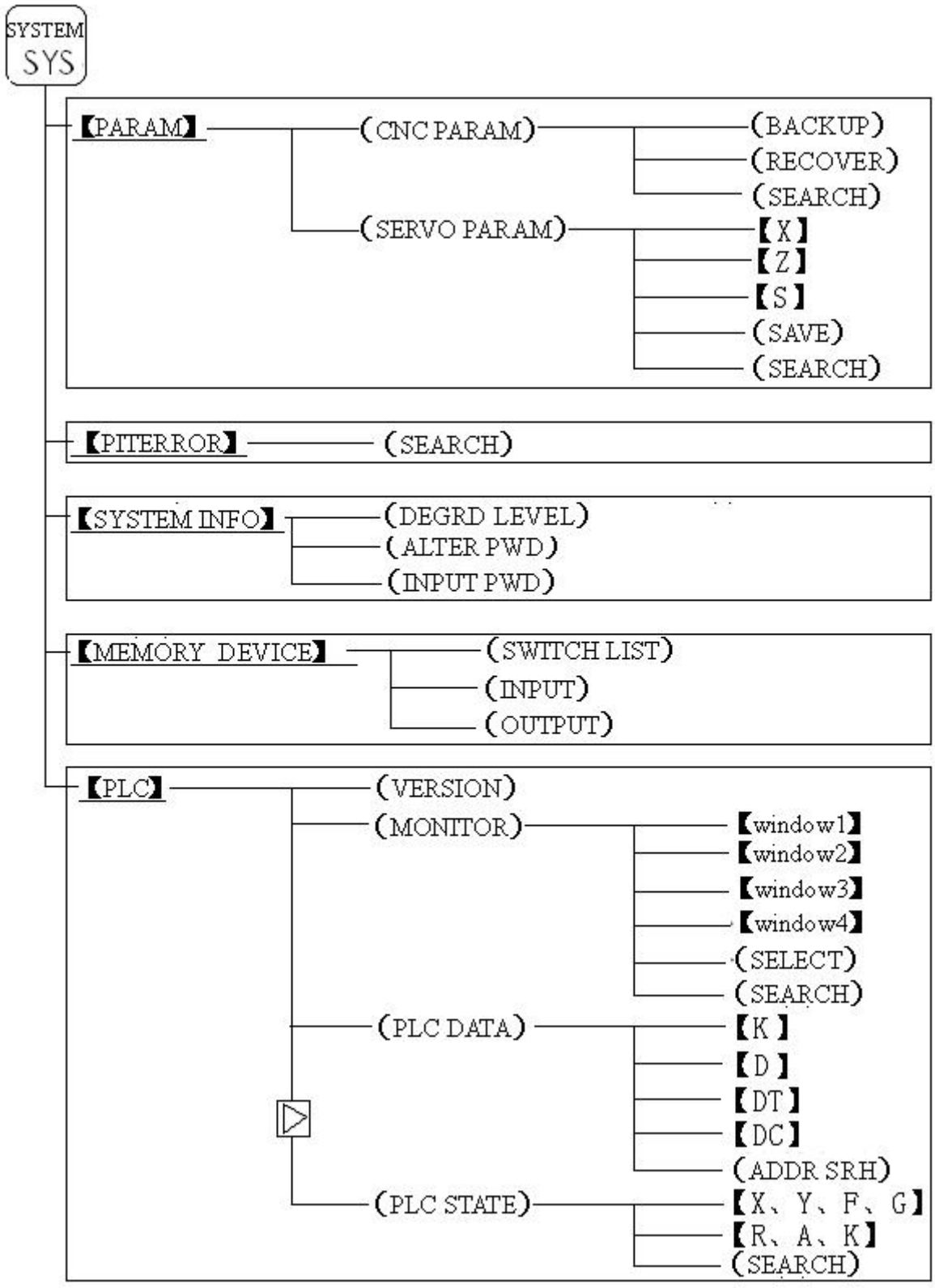
1. **【 】：soft key**
2. **【LOCAL】：switch the windows by the function key POSITION**
3. **（）：soft key in blue font**
4. **：continuous menu key（the first right soft key）**
5. **：key for returning to menu（the first left soft key）**
6. **Some soft keys and their window are not displayed based on the different allocation**



Note: Press (RELATIVE COORDINATE SETTING) key in all position windows, and the system automatically skips the relative coordinate window to execute the relative coordinate setting.

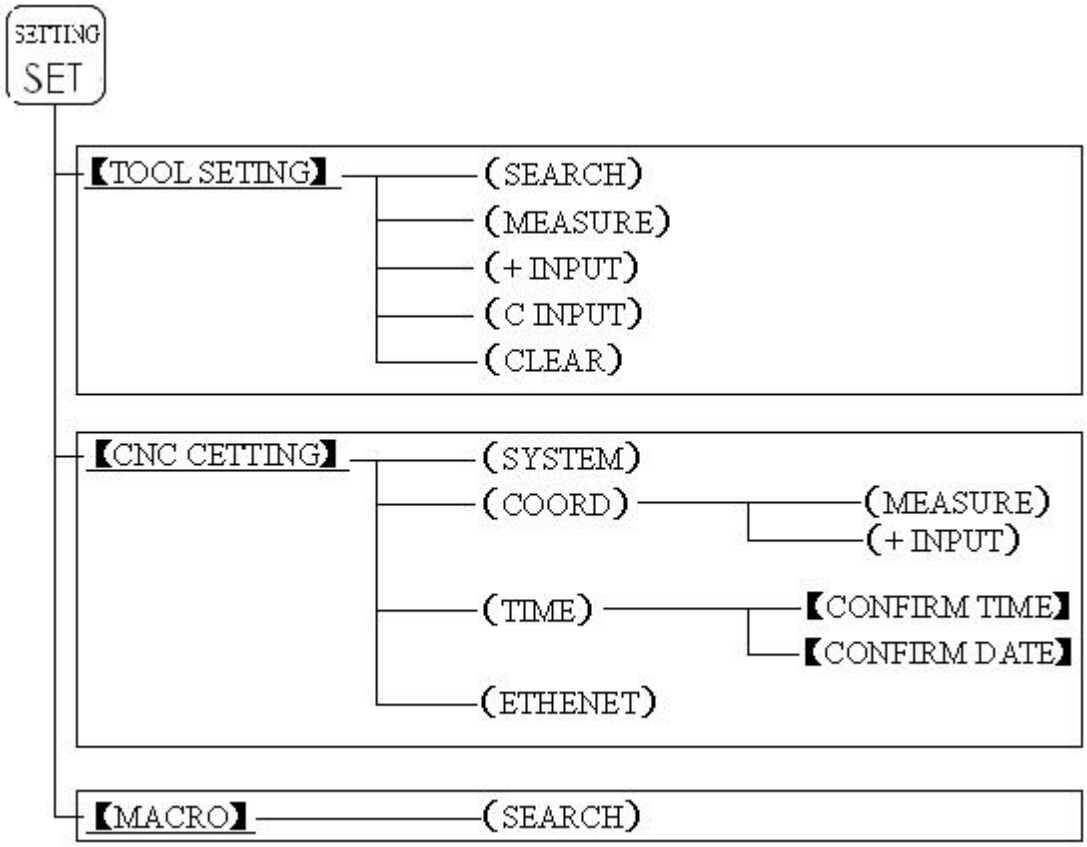


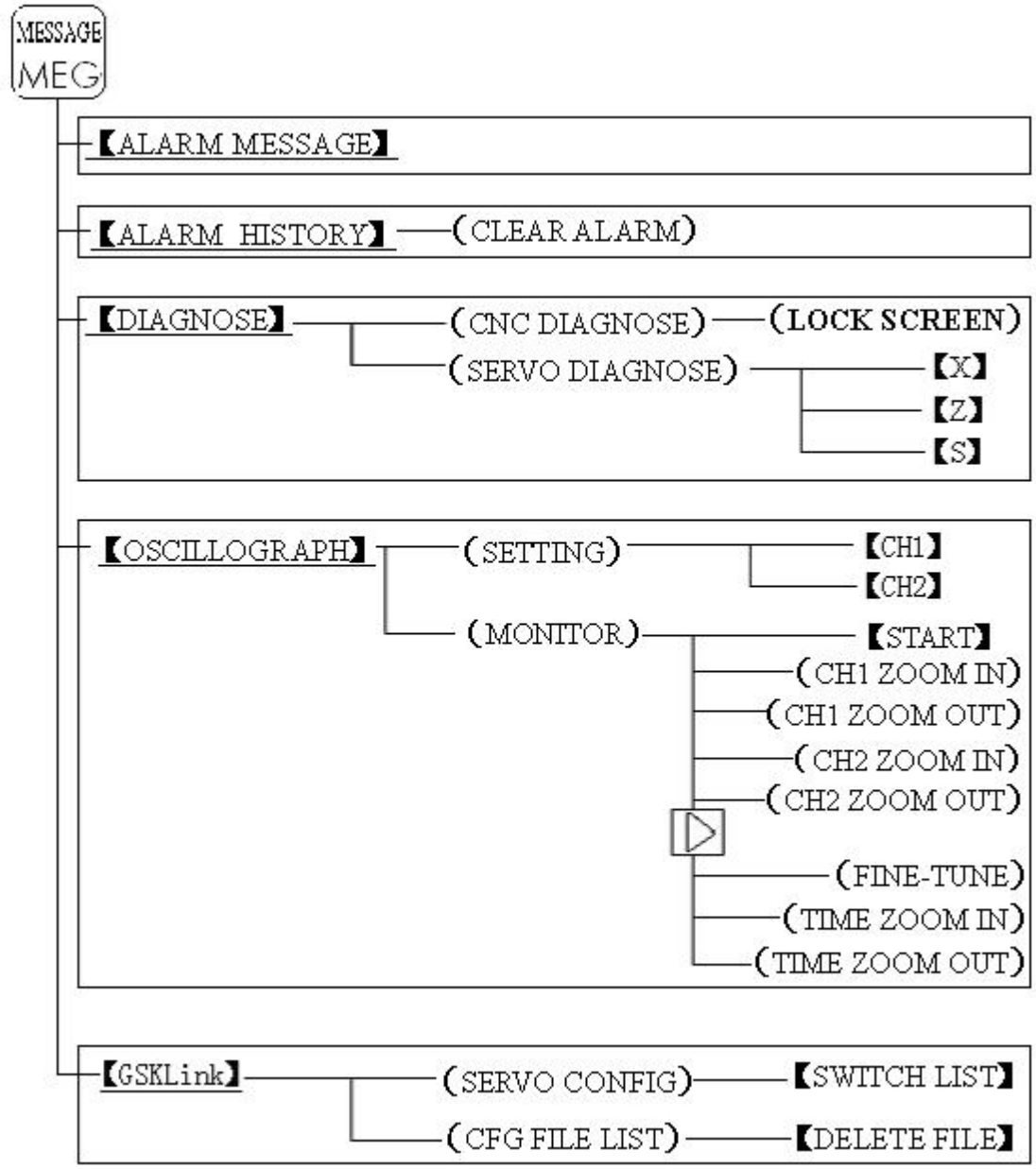
Note: It can be displayed after U disk is inserted in the U disk catalog.



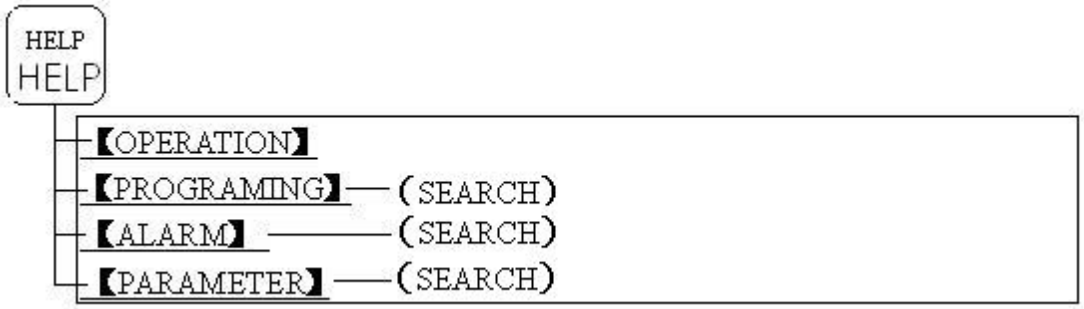
Note 1: The servo parameter is displayed only when the system servo communication function is valid and all servo axes are connected.

Note 2: The operations about the file management are valid only when the U disk is inserted.


SETTING
SET
【TOOL SETING】
(SEARCH)
(MEASURE)
(+ INPUT)
(C INPUT)
(CLEAR)
【CNC CETTING】
(SYSTEM)
(COORD)
(MEASURE)
(+ INPUT)
(TIME)
【CONFIRM TIME】
【CONFIRM DATE】
(ETHENET)
【MACRO】
(SEARCH)



Note: The servo diagnosis is displayed only when the system servo communication function is valid and all servo axes are connected.



3.1 Position Display Window

The initial display is the position window after the system is turned on. Fig.3-1 is the position window display diagram without loaded programs in the state of reset in Auto mode.

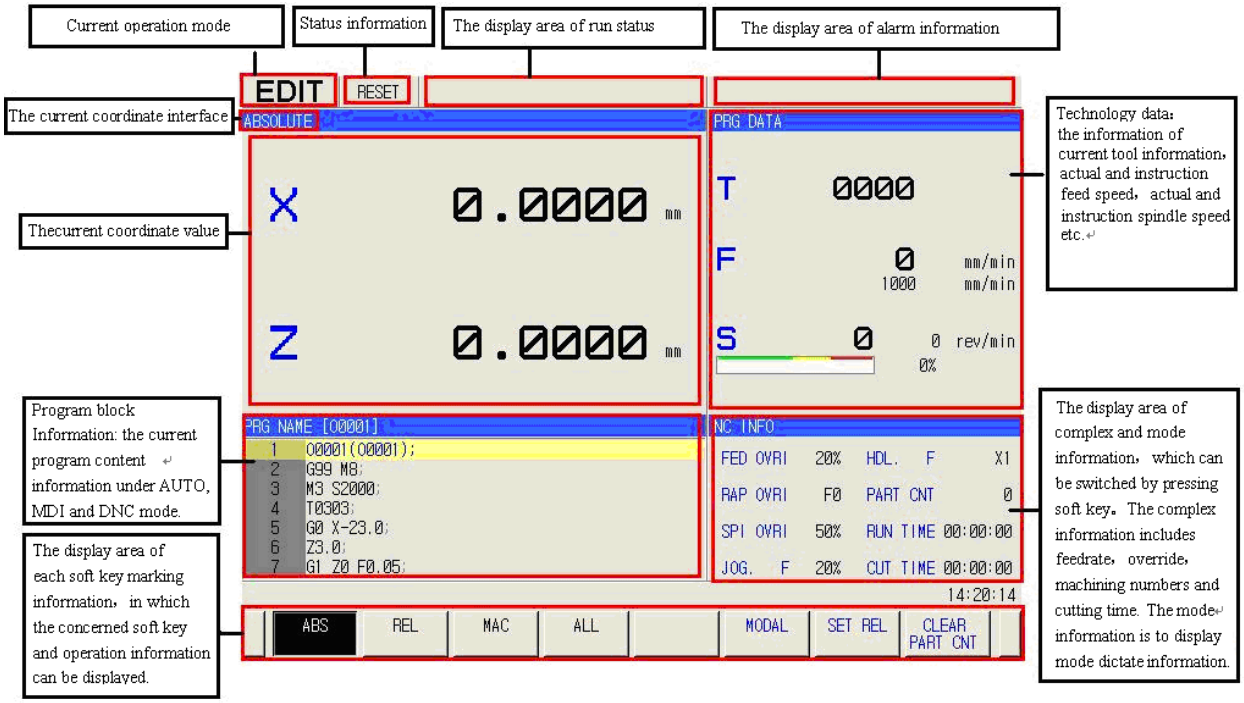

Current operation mode
Status information
The display area of run status
The display area of alarm information
EDIT
RESET
The current coordinate interface
ABSOLUTE
PRG DATA
Technology data: the information of current tool information, actual and instruction feed speed, actual and instruction spindle speed etc.
Thecurrent coordinate value
X 0.0000 mm
Z 0.0000 mm
T 0000
F 0 mm/min
1000 mm/min
S 0 0 rev/min
0%
Program block Information: the current program content information under AUTO, MDI and DNC mode.
PRG NAME [O0001]
1 O0001(O0001);
2 G99 M8;
3 M3 S2000;
4 T0303;
5 G0 X-23.0;
6 Z3.0;
7 G1 Z0 F0.05;
NC INFO
FED OVRI 20% HDL. F X1
RAP OVRI F0 PART CNT 0
SPI OVRI 50% RUN TIME 00:00:00
JOG. F 20% CUT TIME 00:00:00
14:20:14
The display area of each soft key marking information, in which the concerned soft key and operation information can be displayed.
ABS
REL
MAC
ALL
MODAL
SET REL
CLEAR PART CNT
The display area of complex and mode information, which can be switched by pressing soft key. The complex information includes feedrate, override, machining numbers and cutting time. The mode information is to display mode dictate information.

Fig. 3-1

Note: The displayed content is different according to the different allocation, and the diagram form and all content of the related window are based on the standard 2-axis turning machine allocation.

Press POSITION and the system enters the window, and the position window includes the absolute coordinate, the relative coordinate, the machine coordinate and other sub-page shown in Fig.3-2, the corresponding soft key can search all displayed content in each window.

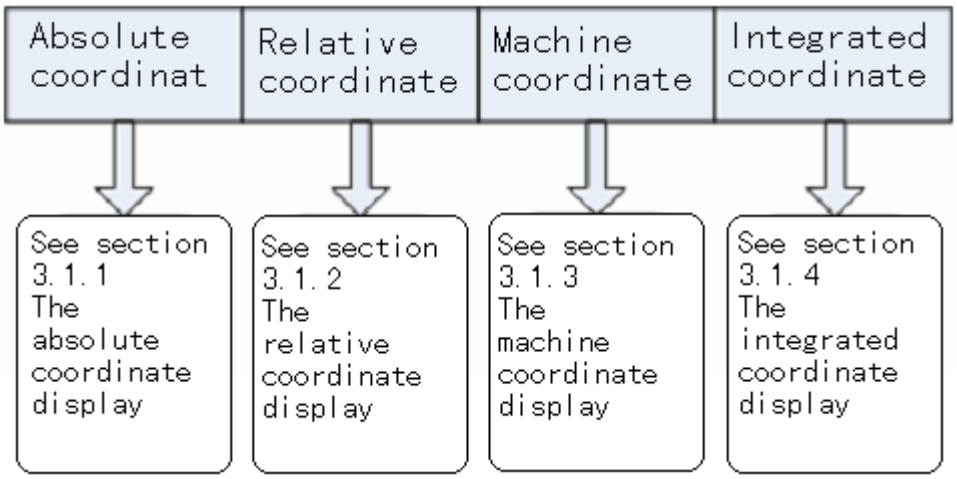

Absolute coordinat
Relative coordinate
Machine coordinate
Integrated coordinate
See section 3.1.1 The absolute coordinate display
See section 3.1.2 The relative coordinate display
See section 3.1.3 The machine coordinate display
See section 3.1.4 The integrated coordinate display

Fig.3-2

3.1.1 Absolute coordinate window

On the position window, press ABS to switch into the absolute coordinate window. During auto mode and resetting, the window is shown as below. On the top left corner, display the coordinate value of X and Z axes as the absolute position of the current work piece coordinate system which the tool is.

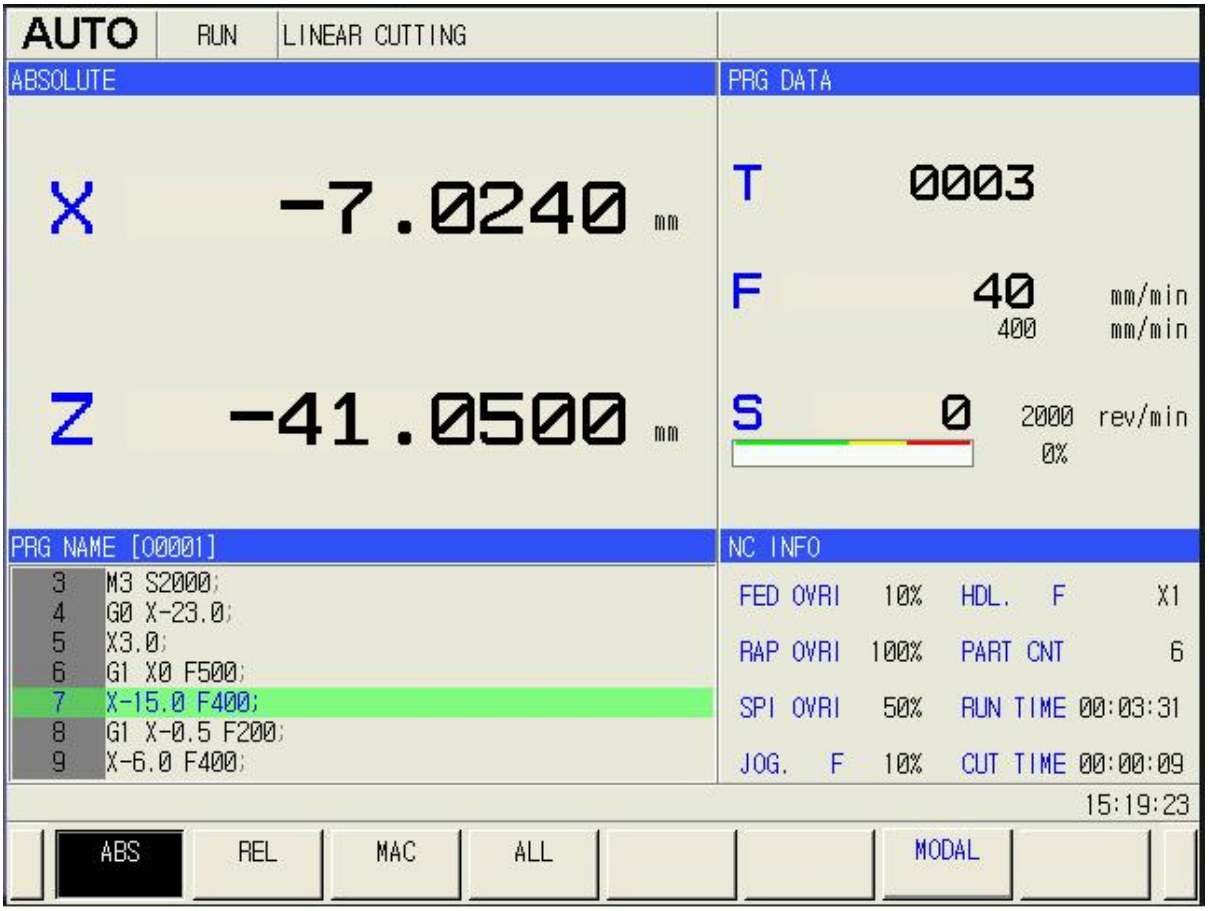


Fig.3-3

The system displays the current operation mode as Auto mode and its state is run at the top left corner. The system prompts LINEAR CUTTING .

The coordinate display area displays X, Z coordinate values as the absolute position of the tool in the current workpiece coordinate system.

Technology data:

T: Current tool number and tool offset number

Actual speed F: During actual processing, the actual processing speed after feeding override;

Programming speed: speed is set by F code in program;

Spindle actual speed S: The spindle speed feed back by the spindle encoder can display the actual speed of the spindle only after installing the spindle encoder;

Programmed spindle speed S: The spindle speed is specified by S code in program;

Comprehensive message:

Feedrate override: It is selected by the feeding override switches;

Rapid override: It is selected by the rapid override switches;

Spindle override: It is selected by the spindle override switches;

Manual override: It is selected by the manual override switches;

MPG override: Current MPG override;

Quantity of processing work pieces: The quantity of the processing work pieces pluses one after the program executes M02 or M30 or M codes set by parameter # 6710.

Cutting time: Executing time of auto running in one time without the time of stop and feeding pause, timing begins from 0 after auto running starts each time, the units in turn are hour, minute and second;

Running time: All execution time of system in auto mode without time of stop and feeding pause is the accumulative cutting time;

G function codes: The mode values of G codes in each group;

Switch between the mode and comprehensive message through pressing MODAL and NC INFO .

Program display area: display the program which is being executed. The block with green is a program which is being executed.

3.1.2 Relative coordinate display

In position window, press REL to switch into the relative coordinate window. Then, on the left top corner, display the relative coordinate value. U and W coordinate value is the relative coordinate value of the current position. U and W coordinates can be cleared during stop and resetting state. The window is shown as below:

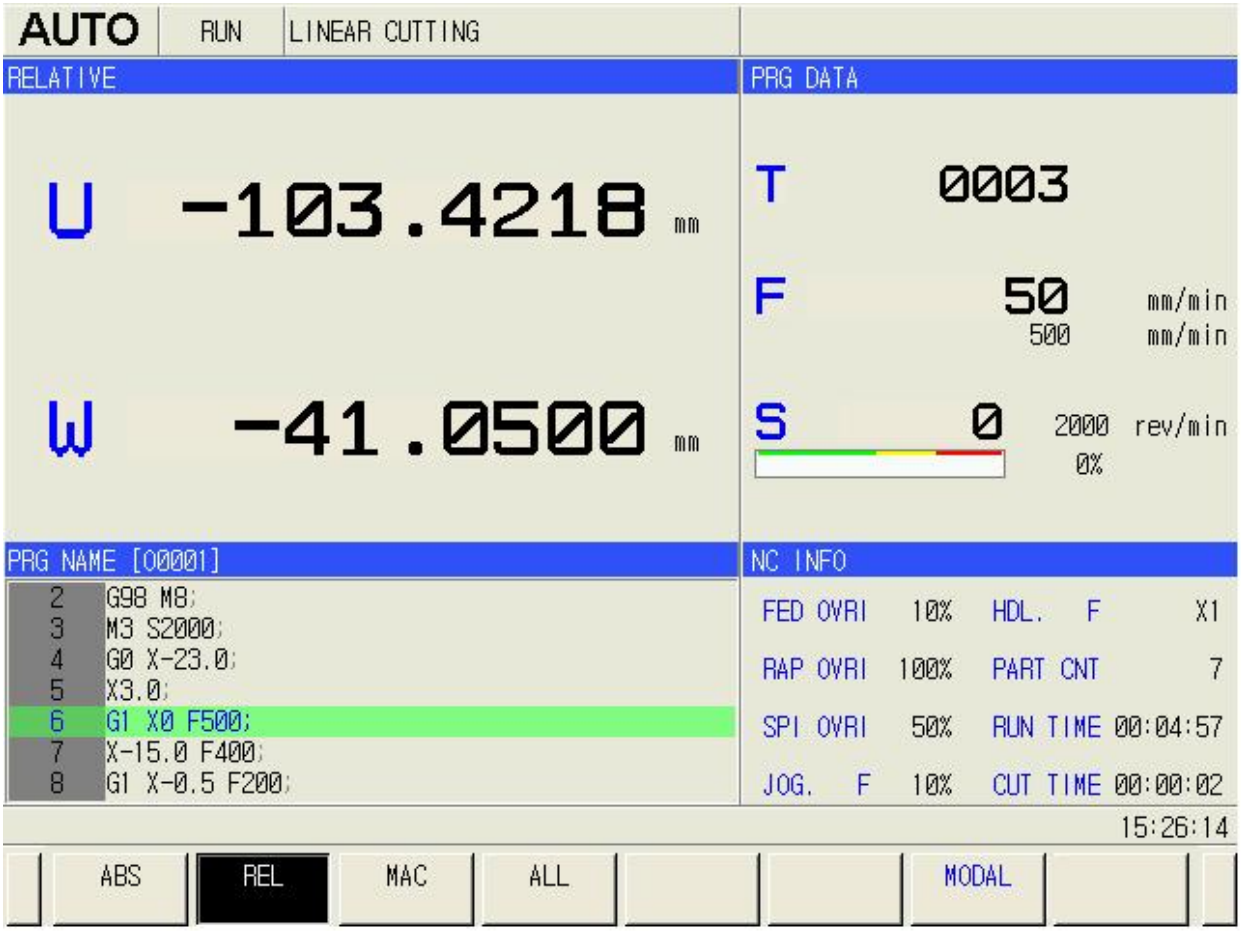


Fig.3-4

3.1.3 Machine coordinate display

On the position window, press MAC to switch into the machine coordinate window. The machine coordinate system is set through the reference position. The window is shown as below:

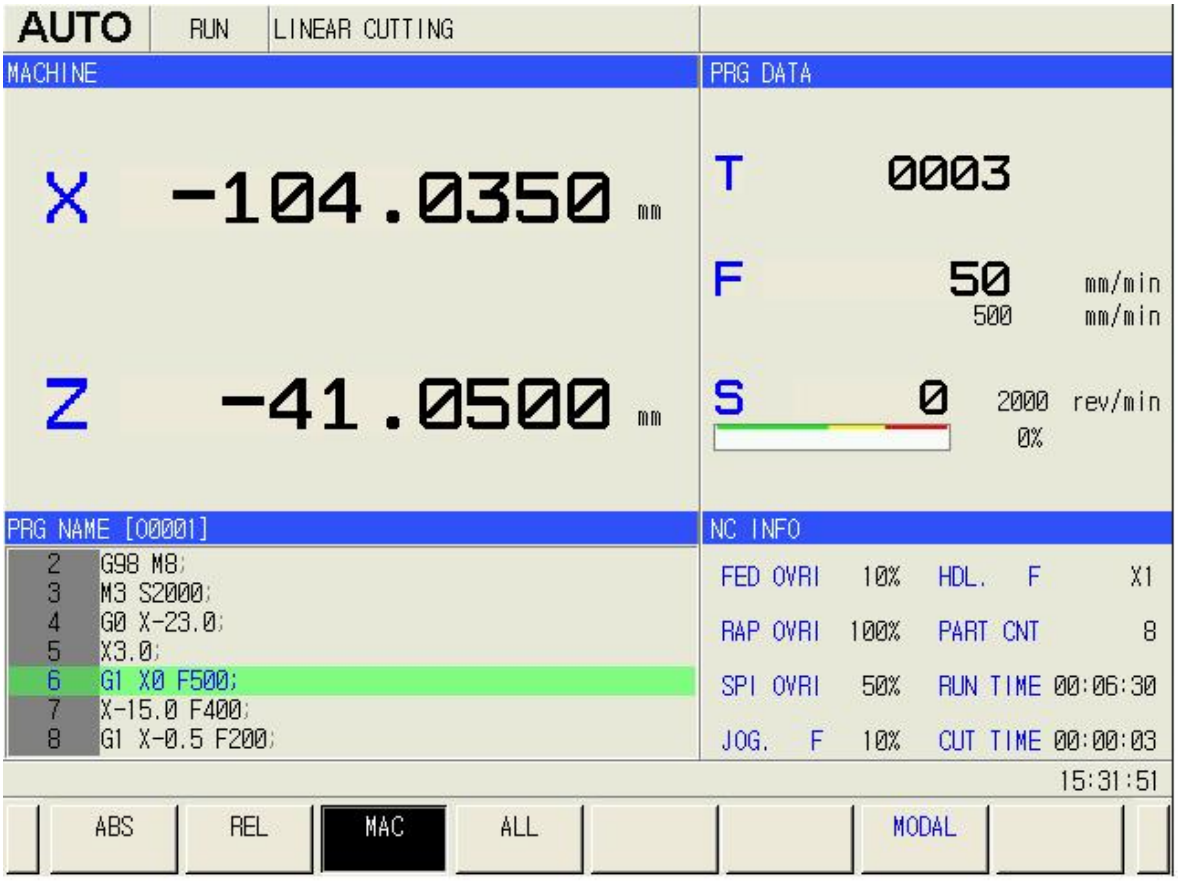


Fig.3-5

3.1.4 Comprehensive coordinate

In position window, press ALL to switch into the comprehensive coordinate window. Then, the comprehensive coordinate value is displayed on the top corner of the window including the absolute, relative and machine coordinates and the surplus movement value. The window is shown as below:

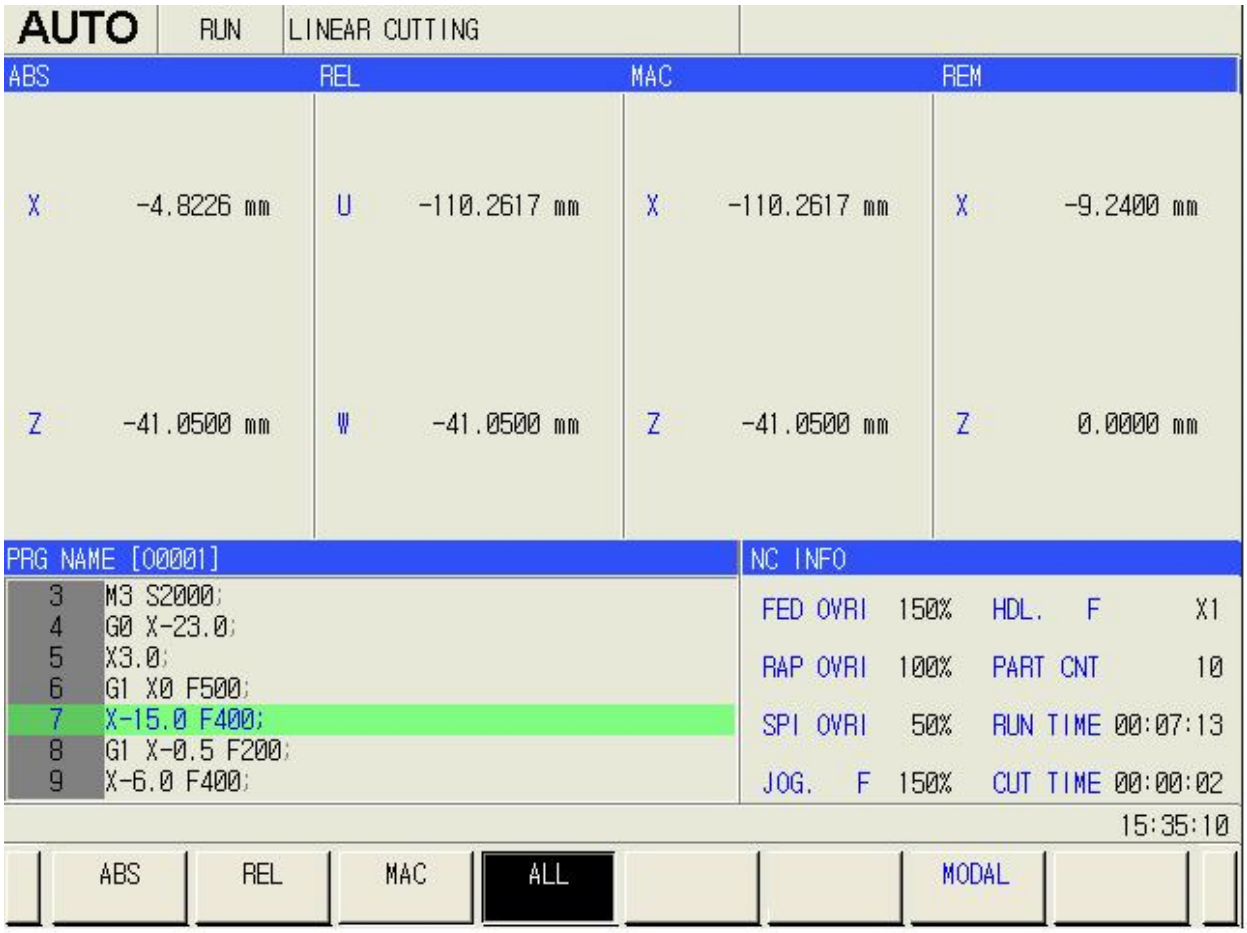


Fig.3-6

3.1.5 Setting the relative coordinate

In position window, press SET REL to set the relative coordinate and the window is shown as below. Then, the relative coordinate value of each coordinate axis can be set. The steps are as following:

（1）During resetting, press SET REL to change the relative coordinate axis into the input state, the relative coordinate value U is shown as below:

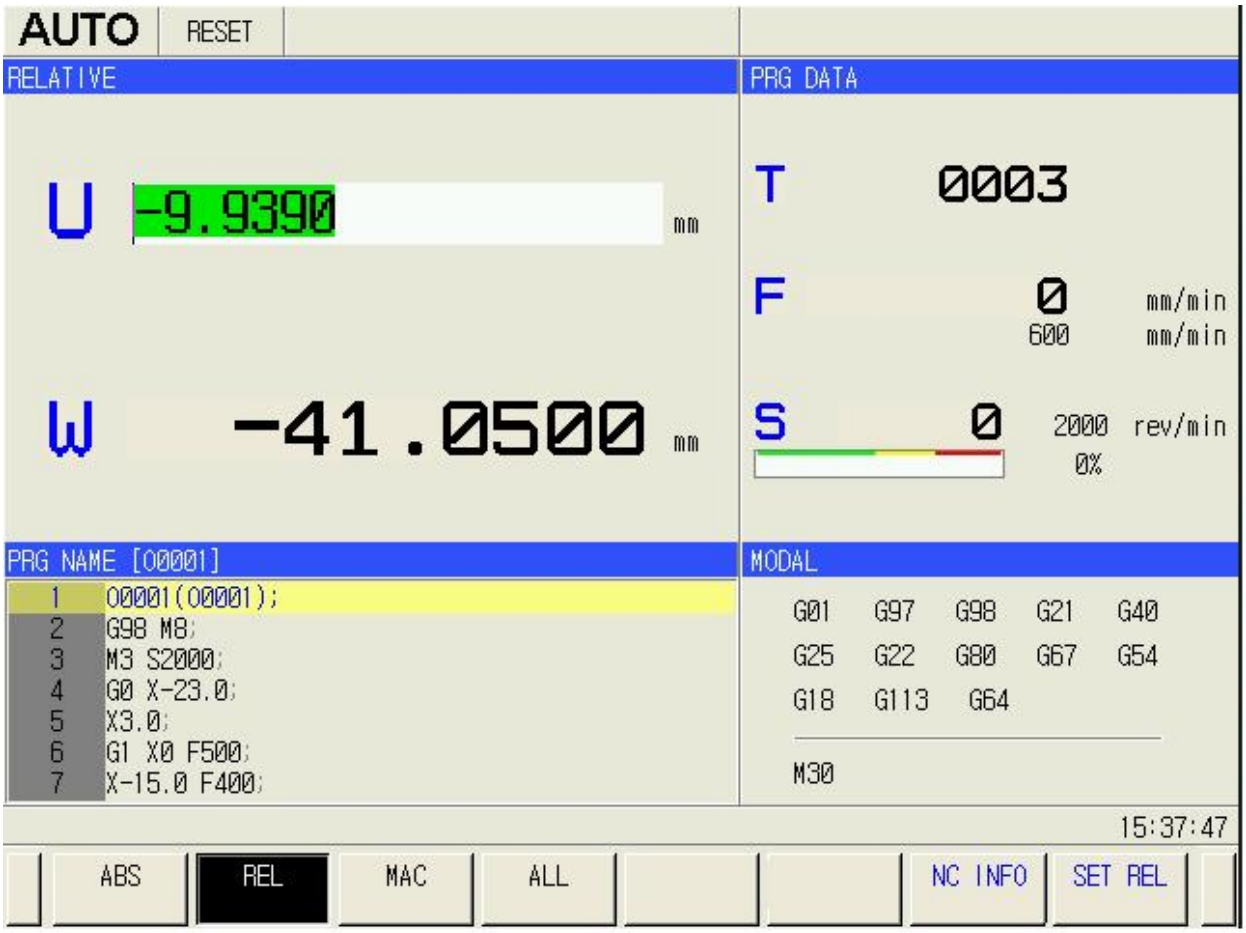


Fig.3-7

（2）Press ⇧ or ⇩ to select the coordinate axis to be set, which makes the axis to be input;

（3）Input the relative coordinate axis to be set, press INPUT to complete setting.

（4）Firstly press SET REL , and then press ⇧ or ⇩ to select the other coordinate axes to set the relative coordinate value.

3.1.6 Switching between the mode and the comprehensive message

In position window, press MODAL and NC INFO to switch between the mode and the comprehensive message, the mode window is shown as below:

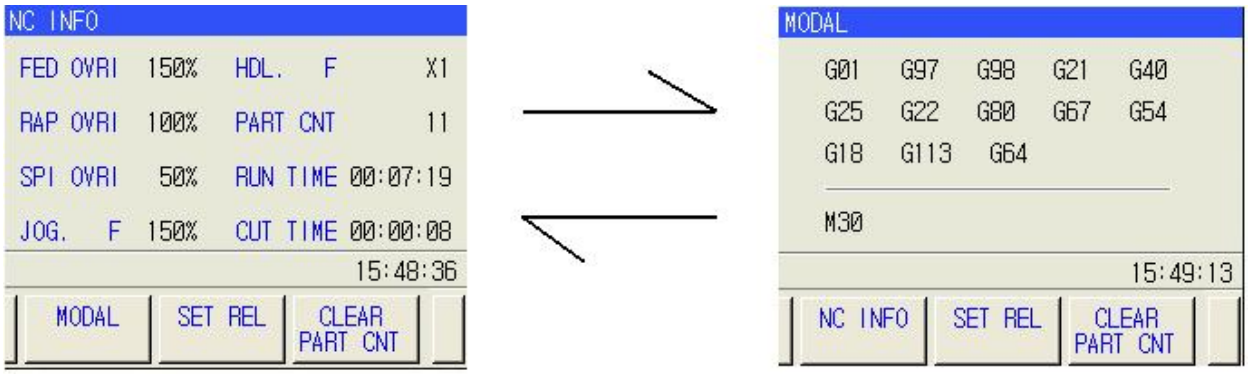


Fig.3-8

3.1.7 Clearing workpiece count

In the position window, press CLEAR PART CNT to clear the currently machining workpiece count, and the mode display window is shown in Fig.3-9:

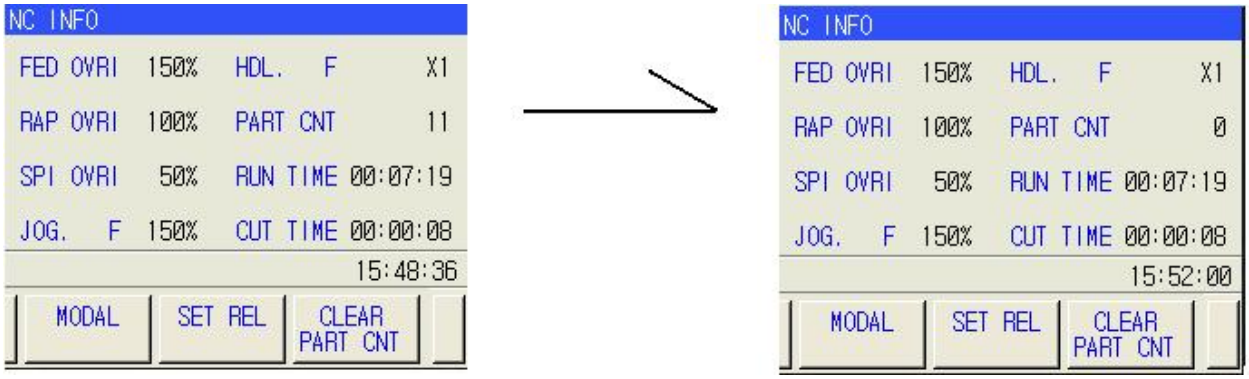


Fig.3-9

3.1.8 Clearing run time

In the position window, press CLEAR RUNTIME to clear the currently machining workpiece count, and the mode display window is shown in Fig.3-10:

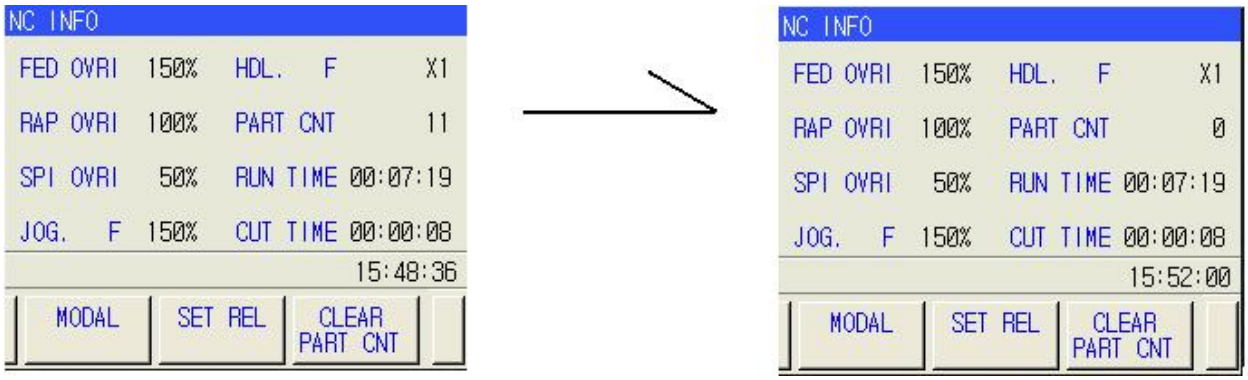


Fig. 3-10

3.2 Program Window

Press PROGRAM to enter the program window including the local directory, MDI program, item/times display, and also the U disk directory is displayed when the U disk is inserted.

3.2.1 Local directory and U disk directory

- **Local directory**

Press LOCAL to load, open, copy, paste, create, save as, delete, rename, search and other operations for the programs in the local directory.

The local directory in the program is shown in Fig. 3-10:

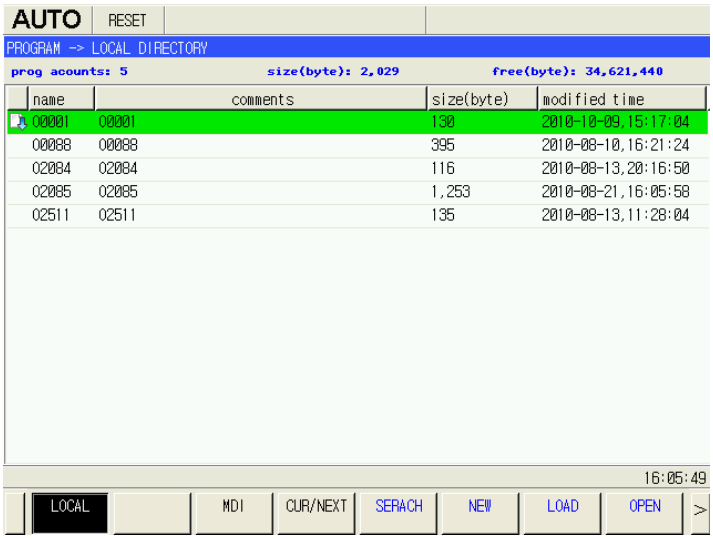


Fig. 3-10

The run mode and the state of the system are displayed in the top state message display area of the window; the program count, the occupied capacity of all programs and the left capacity of the current system are displayed in the below,

In the list, the program list and each program size and the recently modified Value are displayed in the current system. The program in the blue is the one selected by the current cursor as the program O0005 shown in the above-mentioned figure. The program in the red with the note is currently loaded to the position display window and can be executed, such as the program O0000 shown in Fig.3-10.

- **U disk directory**

When the system USB window has the U disk, and simultaneously displays one “U disk directory” soft key as Fig.3-11. press USB and the window displays CNC program directory in “NCPROG” file in the U disk. The input, output and other operations to the files in the U disk can be executed.

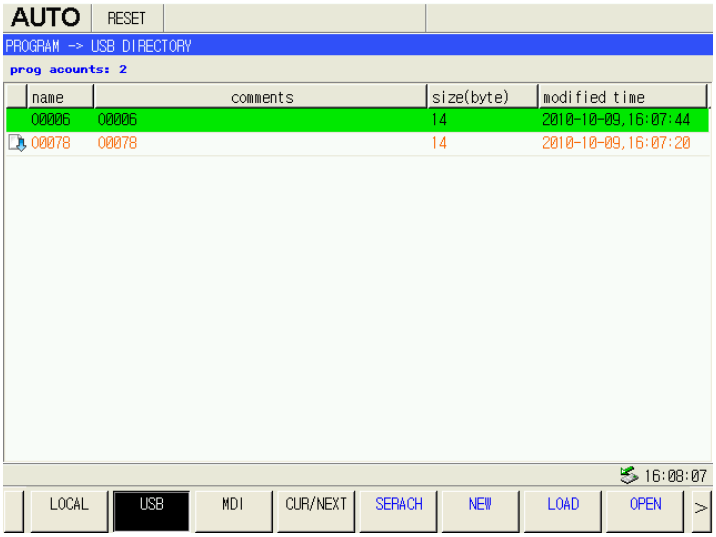


Fig. 3-11

Note: Others related to the program window and its window are referred to the Chapter VI.

3.2.2 MDI program

Press PROGRAM to enter the program window, press MDI to display MDI program input box, G, F, S mode, and the executed M command. In MDI working mode, the most 10 lines of the NC program can be input in MDI input box shown in Fig. 3-12:

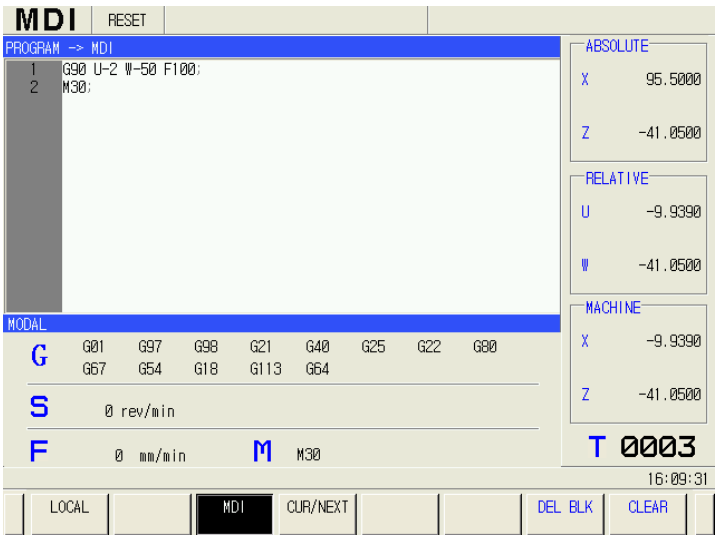


Fig.3-12

In MDI working mode, the system can display the soft key DEL BLK and CLEAR. Press DEL BLK to delete the NC command where the cursor is. Press CLEAR and all NC commands in MDI input box are cleared.

3.2.3 Item/times

Press PROGRAM to enter the program window, press CUR/NEXT to display the current block being executed and the NC command of the next block shown in Fig. 3-13:

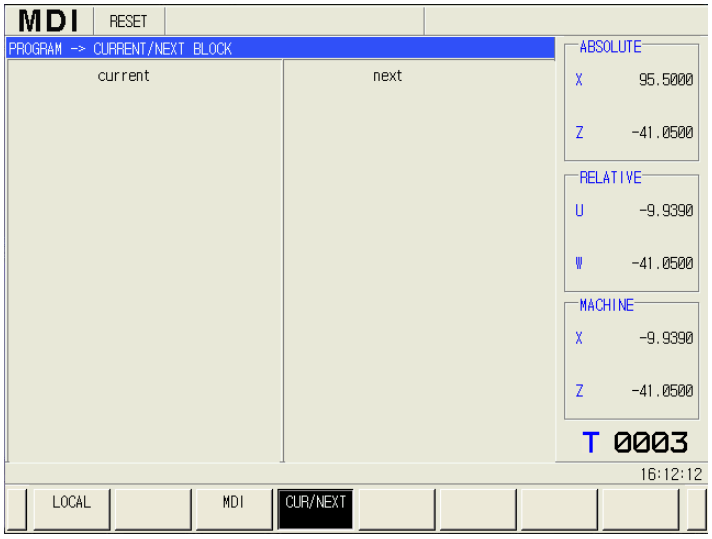


Fig. 3-13

3.3 System Window

Press SYSTEM to access the system window. It mainly includes windows of parameter, screw pitch compensation, system message, file management and ladder diagram, etc. Check the content in each window through the corresponding soft keys, and the structure of the soft key is shown as below:

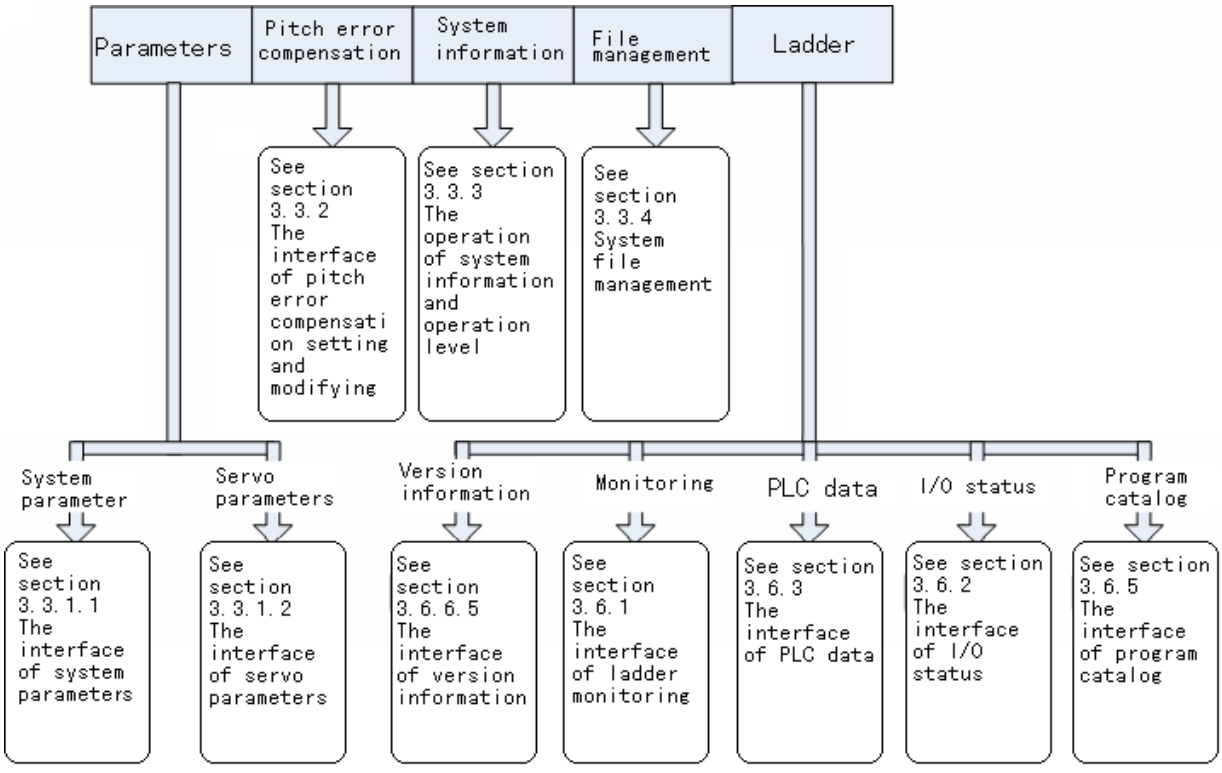

Parameters
Pitch error compensation
System information
File management
Ladder
See section 3.3.2 The interface of pitch error compensation setting and modifying
See section 3.3.3 The operation of system information and operation level
See section 3.3.4 System file management
System parameter
Servo parameters
Version information
Monitoring
PLC data
I/O status
Program catalog
See section 3.3.1.1 The interface of system parameters
See section 3.3.1.2 The interface of servo parameters
See section 3.6.6.5 The interface of version information
See section 3.6.1 The interface of ladder monitoring
See section 3.6.3 The interface of PLC data
See section 3.6.2 The interface of I/O status
See section 3.6.5 The interface of program catalog

Fig.3-14

3.3.1 System parameter setting and rewriting window

On the system window, press PARAMETER to access parameter setting window, which is shown in Fig.3-15:

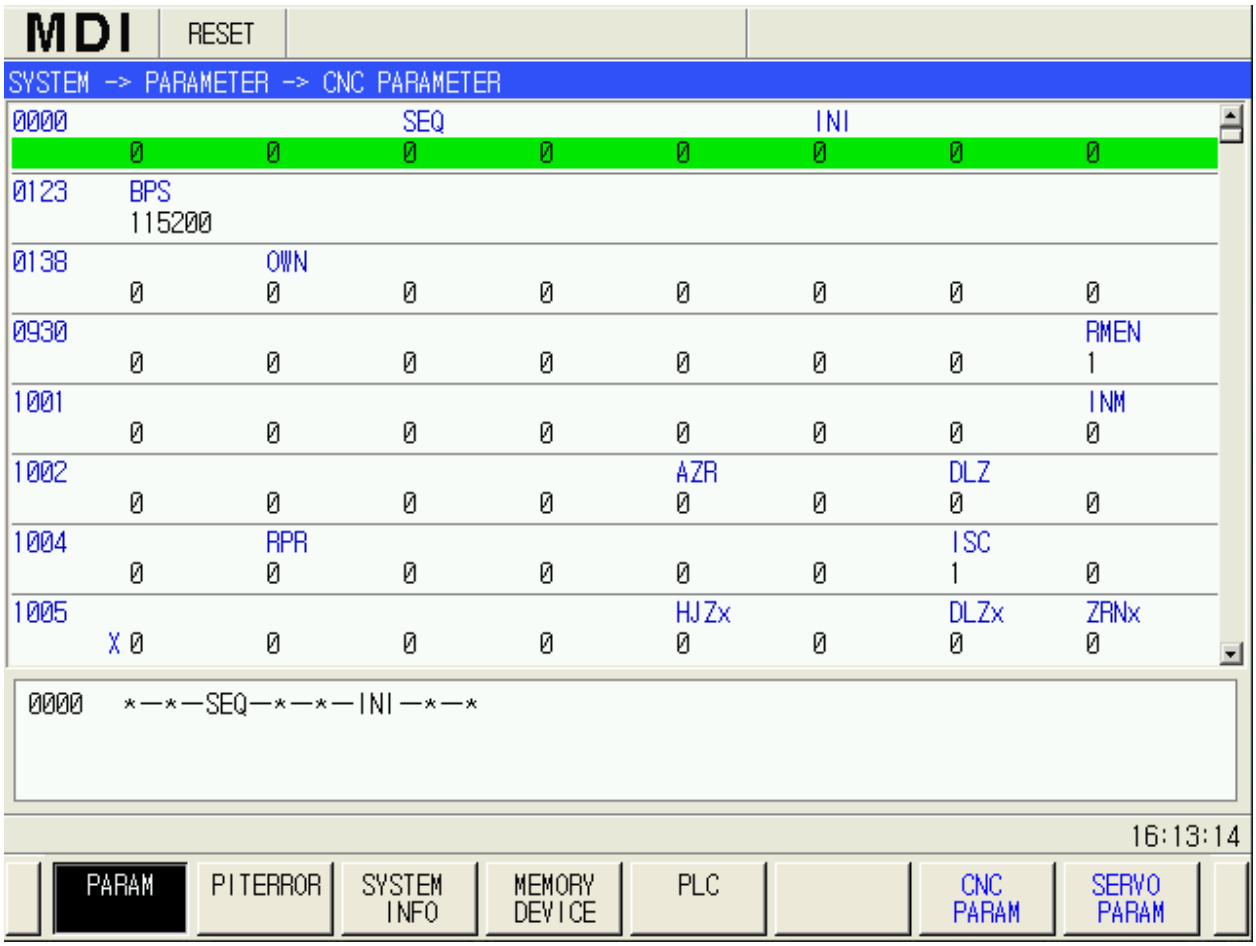


Fig.3-15

3.3.1.1 System parameter window

On the system window, press PARAMETER and CNC PARAM to access the system parameter setting window.

The window displays the detailed message of the user parameter, set and rewrite the system parameter in the window, back up the parameter set currently, and initialize the parameter default by the system or the parameter of user backup.

In MDI mode, when the parameter switch is on and the operation authority is above level [3], the parameter can be set. Press , , or to select the parameter to be rewritten; or press SEARCH to input the parameter sequence number to be selected; press OK and the cursor positions in the parameter, like parameter #0000, which is shown as

above; press INPUT, the parameter can be rewritten, parameter #0000 is shown as the following figure:

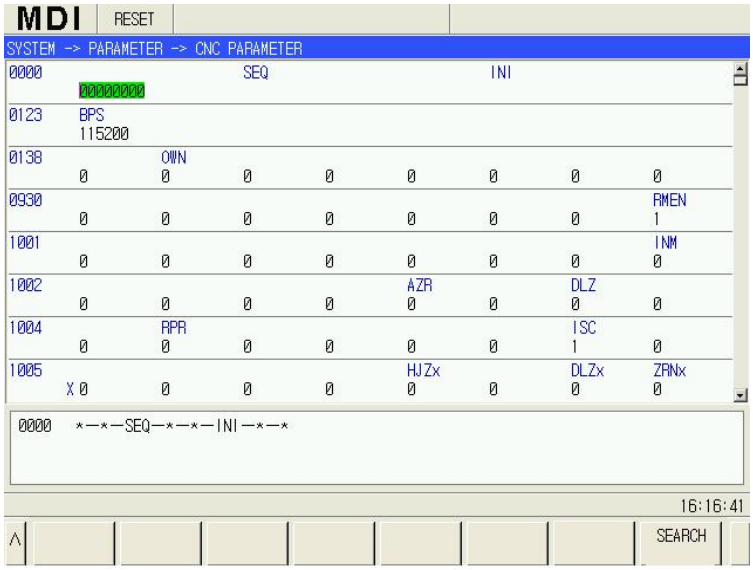


Fig.3-16

Press the numerical key to rewrite the value of 8 bits in binary system, and press INPUT to confirm the setting is completed; if the value is less than 8 bits, zeroing in upper bit;

Moreover, set bit parameter based on the bits;

（1） In parameter setting window, press , , or to select the parameter to be set.

（2）Press or to select the parameter bit to be rewritten.

（3）Repeatedly press INPUT the parameter bit is switched between 0 and 1, and the value of the parameter bit is rewritten.

（4）Press , , or to select the other parameters to be set.

The numerical parameter setting method is similar to that of the bit parameter:

（1）Using , , , selects the required parameter to modify; or press SEARCH to input the sequence number of the selected parameter, and then press OK, and the cursor positions to the parameter.

（2）Press INPUT to make the selected parameter to be modified.

（3）Input the set value and press INPUT to confirm the setting.

（4）Using , , , selects other parameters to be set.

Note 1: After rewrite the system parameter, some parameter can become valid immediately, some parameter becomes valid after the system powers on again, refer to 988T parameter introduction.

Note 2: Only in MDI mode, when the parameter switch is on and the operation authority is above level (3), the parameter can be set and rewritten.

3.3.1.2 Servo parameter window

On the system window, press PARAMETER and SERVO PARAM to access the servo parameter setting window.

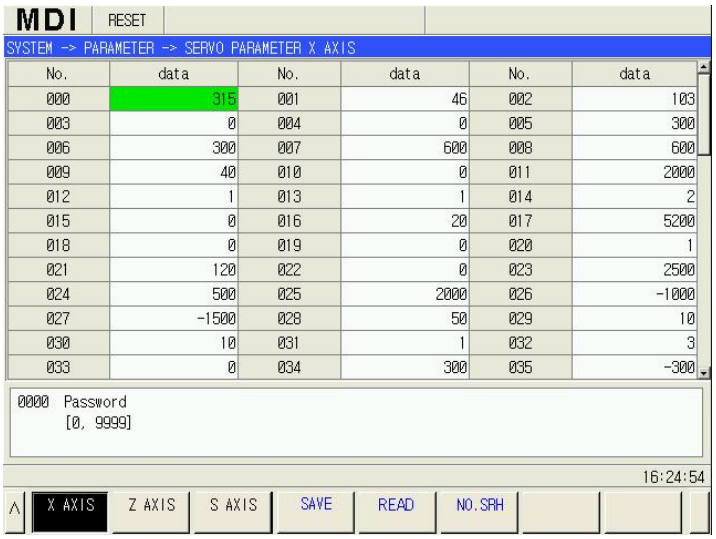


Fig.3-17

The servo parameter window mainly includes checking the servo parameter and rewriting and saving the servo parameter in CNC side.

Switching axes: Press X AXIS, Z AXIS and S AXIS to switch the servo parameters among X, Z and S axes.

Rewriting a parameter: Press INPUT to input the parameter value; press INPUT again to complete the rewriting.

Searching a parameter: Press NO.SRH to input the parameter number, and the operation is completed.

Saving a parameter: After rewriting the servo parameter, press SAVE to save the rewritten parameter value after servo power off.

Note 1: Before using, the servo system should be connected correctly and the configuration of the servo slave number should be right.

Note 2: Only in MDI mode, when the parameter switch is on and the operation authority is above level [3], the parameter can be set and rewritten.

Note 3: The motor's default parameter cannot be modified at will, if done, please contact with us.

3.3.2 Screw pitch compensation setting and rewriting window

On the system window, press PITERROR to access the screw pitch compensation window, which is shown as below:

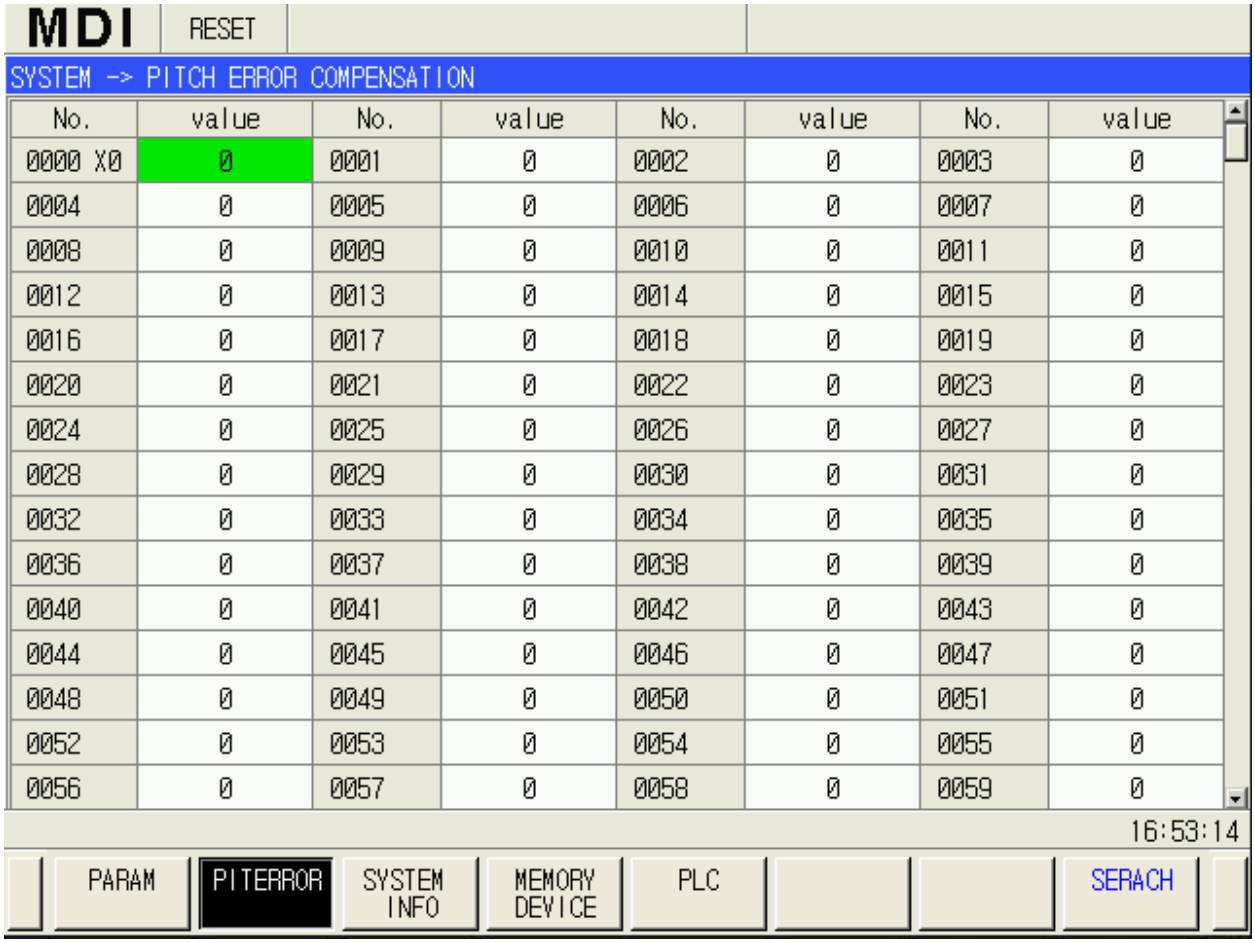


Fig.3-18

On the window, the user can check and set the screw pitch compensation value corresponding to each screw pitch number.

On the screw pitch compensation window, press or and , , or to select the compensation value of screw pitch compensation number to be set, or press SERACH to search for the screw pitch compensation number and the cursor positions to the compensation value of screw pitch compensation to be rewritten.

When the operation authority is above level (2), press INPUT and the compensation value of the screw pitch compensation number can be rewritten, the compensation value #0000 is shown as below:

3.3.3 System message and operation authority levels

On the system window, press SYSTEM INFO to access the system message window, which is shown as below:

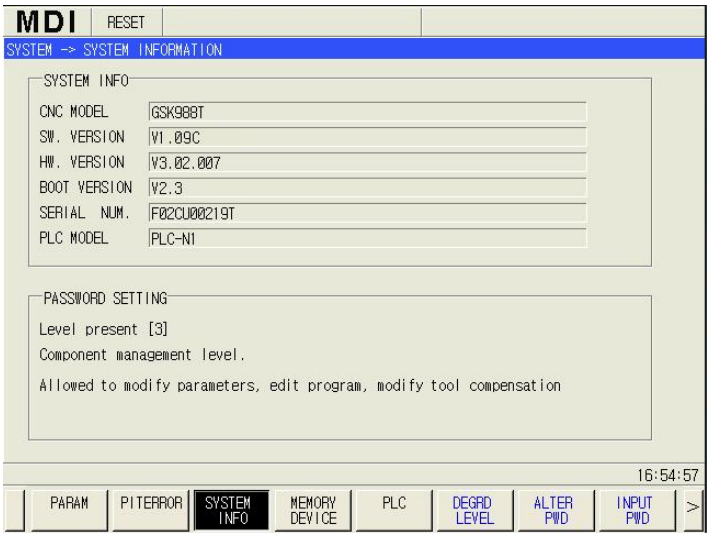


Fig.3-19

On the system message display window, it mainly displays the product type, software, hardware and BOOT versions, system serial number, PLC modal and the operation authority. On the window, the operation authority level password can be rewritten and the operation authority level can be set, etc.

To realize the multi-level operation authority management of the development, maintenance, machine design and equipment management, etc, GSK988T CNC system sets operation authority of 5 levels, 1 is the superlative,5 is the lowest;

Level 1: Development with system software maintenance authority;

Level 2: Machine manufacturer with the authority of PLC program editing, screw pitch error compensation Value input and switch off the machine in limited time;

Level 3: (User) equipment management with the authority of rewriting the parameter, editing the part program and the tool compensation Value;

Level 4: Machine operation level with the authority of editing the tool compensation Value and selecting the part program (namely: operate the tool-setting, select the part program of auto running), but the parameter can't be rewritten and the part programs can't be edited;

Level 5: Operation limit level, without operation password (the operation password is canceled), the parameter can't be rewritten, the tool compensation Value can't be edited, and the part program neither be selected nor edited (namely, the tool-setting is invalid, only run the current part program), manual, MPG, zero-return, MDI running and auto running can be operated, the part files of the system can back up rather than download.

Note: Upload means uploading the files of CNC to PC and download means downloading the files to CNC.

The list of operation function relative with the operation authority levels:

Operation function	Operation authority level				
	Level 1 (Development)	Level 2 (Machine manufacturer)	Level 3 (Equipment management)	Level 4 (Machine operation)	Level 5 (limited operation)
System software upgrade	OK	NO	NO	NO	NO
Set the limited time of the system auto off	OK	OK	NO	NO	NO
PLC program editing, downloading and uploading	OK	OK	NO	NO	NO
Input the screw pitch error compensation Value and download the screw pitch compensation file	OK	OK	NO	NO	NO
Upload and download the part program	OK	OK	OK	NO	NO
The parameter switch on (Allowable rewriting the parameter)	OK	OK	OK	NO	NO
The program switch on (Allowable editing the program)	OK	OK	OK	NO	NO
Set tool lift and download its files	OK	OK	OK	OK	NO
Input the macro variable	OK	OK	OK	OK	NO
Input the tool compensation Value (allowable tool-setting) and download the tool compensation and the tool offset files	OK	OK	OK	OK	NO
Upload the screw pitch compensation files	OK	OK	OK	OK	OK
Upload the tool life file	OK	OK	OK	OK	OK
Upload the tool compensation and the tool offset files	OK	OK	OK	OK	OK

If execute the operation limited by the authority level, the corresponding authority must be obtained. Press SYSTEM on GSK988T panel to access the system window, and then press SYSTEM INFO to access the password window, finally press DEGRD LEVEL , ALTER PWD or INPUT PWD to

access the corresponding setting, and input the password corresponding to the operation level, the relative authority is obtained. On the password setting window, the password of the level or lower than the level can be rewritten, and the current password level can be degraded.

The operation authority of level 1 isn't saved after power off, and access level 2 after power on, again. The operation authority of levels 2～5 are saved, restore the operation authority level after power on again.

When execute the operation which doesn't reach the authority level, it reminds the current operation authority isn't enough in lower left; in auto mode, when the operation level isn't enough, the machine stops moving and alarms.

（1）Access the authority level

Press DEGRD LEVEL to degrade the operation authority level, display the current operation authority level in the authority box.

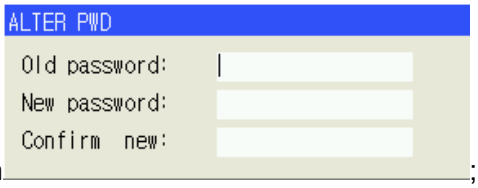
When INPUT PWD is pressed to input the password corresponding to the level to access the level operation authority.

Note: The corresponding relation between the initial password relative to each authority level is shown as below:

Operation levels	Initial password
Level 1	***
Level 2	***
Level 3	333333
Level 4	444444
Level 5	Without password

（2）Rewriting the password

Firstly, access the operation authority level to rewrite the password; press ALTER PWD to rewrite the system authority register password, the window is shown as below:



Input the old and new passwords in ; and press CHANGE to switch between the new and old passwords. Finally, press OK to complete the rewriting password.

3.3.4 System file management

On the system window, press MEMORY DEVICE to access the file management window. The window is shown as below:

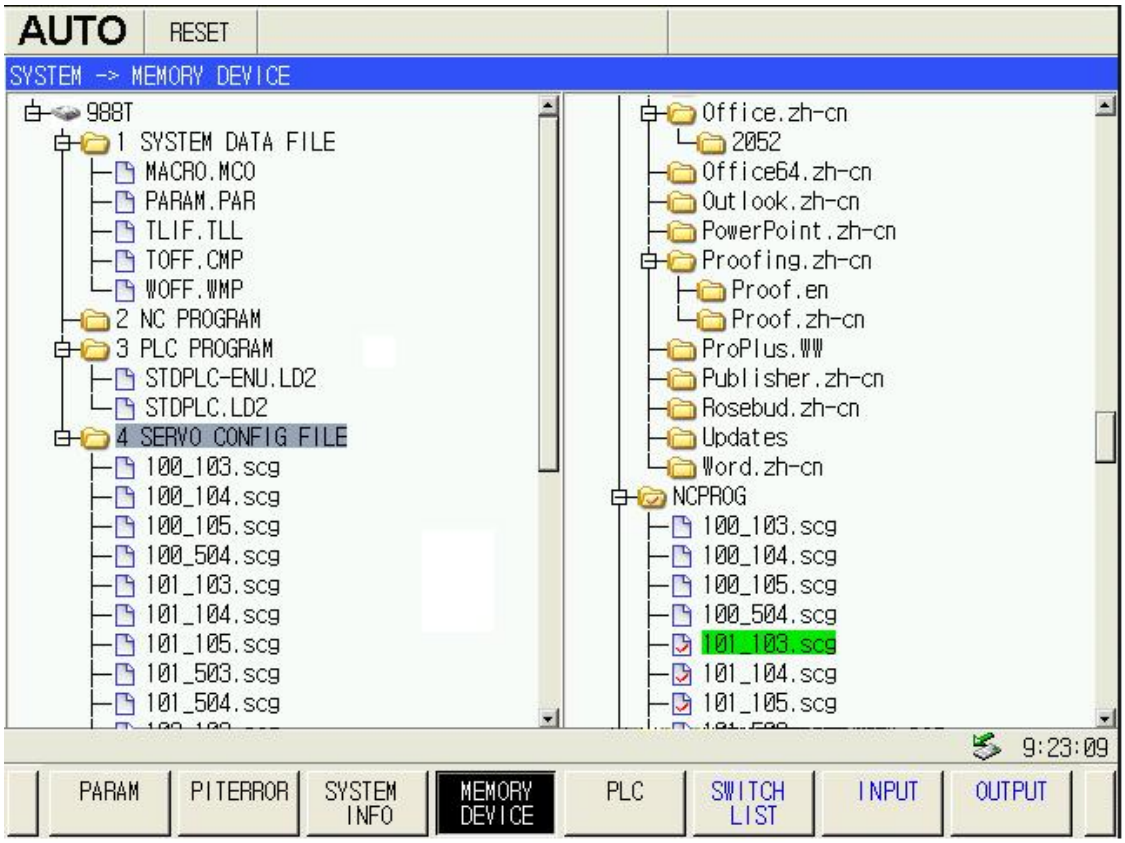


Fig.3-20

The window is divided into left and right columns. The left column displays the system files and the part program file directories; when the system is with the flash disk, the right column displays the file directory in the flash disk, which is shown as the following figure. Then, input or output the system files, the files in the system can be output to the flash disk, or the file in the flash disk can be input into the system.

（1）Press SWITCH LIST and the cursor can switch between the system file directory in left column and the file directory of flash disk in right column.

（2）When the cursor is on the file, press ⇦ or ⇨ to open or close the file.

（3）Press ⇧ or ⇩ and move to the document to be operated, press INPUT to select the document, the selected document is ticked, such as the part programs O0098, O0003 and O0777 in the system file directory, which is shown as above. When the cursor is on the file, then, press INPUT to select all documents in the file.

（4）Then , after select the files in the system, press INPUT to output all the selected files to the

flash disk; After selecting the files in the flash disk, press INPUT to input all the selected files in the flash disk to the system file directory.

3.3.5 Ladder diagram

Because there are too many windows about the ladder diagram, it is introduced independently, about the ladder diagram windows refer to chapter 3.6.

Press SYSTEM, and then press PLC to enter the current PLC display window and to real-time search PLC conditions, the ladder window mainly includes the version message, monitoring, PLC Value, PLC state and other sub-window which content are searched by pressing the corresponding soft keys shown in Fig.3-21.

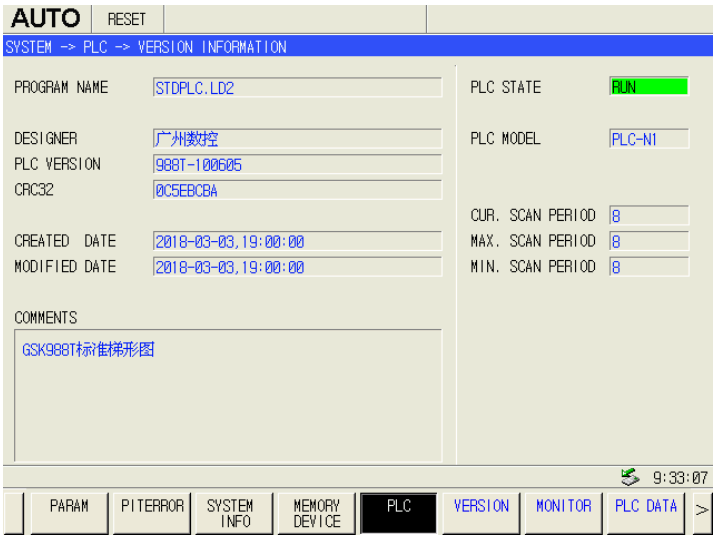


Fig.3-21

The top in the window displays the current run mode and the state; displays the ladder version message, the ladder program of the current run, its run state and others.

3.3.5.1 Ladder monitoring display

Press SYSTEM, and then press the ladder soft key to enter the ladder window, and press MONITOR to enter the run monitor display window of the current PLC shown in Fig.3-22.

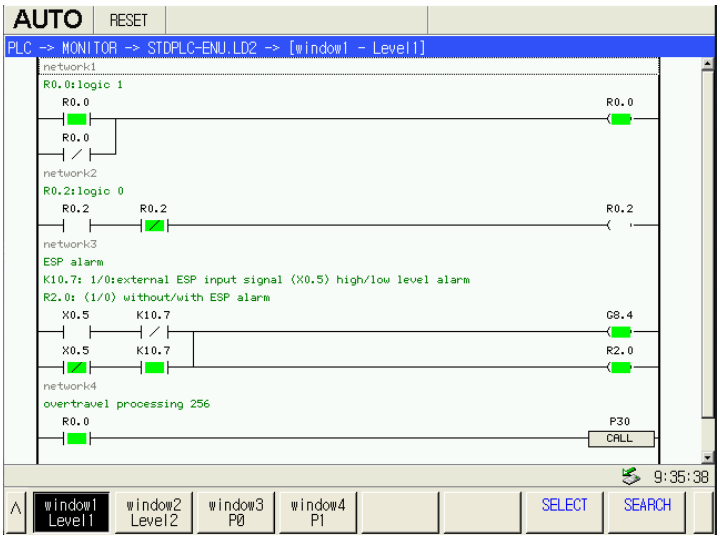


Fig. 3-22

The user can search the ON/OFF of the current contact, the coil in the monitor window, the current value of the timer and the count. The displayed bottom color is blue when the contact and the coil are ON, and it is opposite to the window color when they are OFF. For example, X0.5 ⊣■⊢ means the contact X0.5 ON, Y25.2 ⊣( )⊢ means the coil Y25.2 OFF.

1. Search the window program

In monitor window, the system can monitor the programs of the four window, and the user can press window1 Level1, window2 Level2, window3 P0, window4 P1 to separately search the corresponding ladder block of each window, at this time, the system correspondingly displays the ladder of the block of the correspondingly selected window.

2. Select the window block

（1） Select the window which is required to select the block, i.e. separately press window1 Level1, window2 Level2, window3 P0, window4 P1 to select the windows.

（2）Press SELECT to select the window program, at the time, the display is shown in Fig. 3-23:

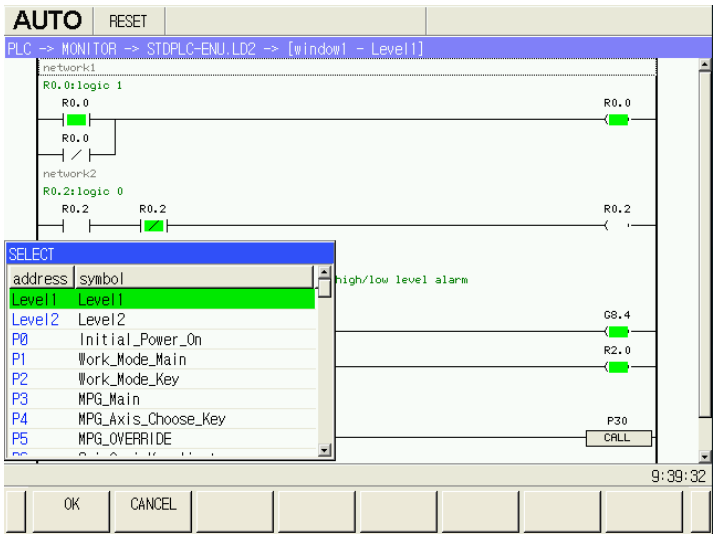
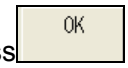
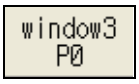
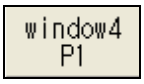


Fig. 3-23

（3）Press , , , to select the ladder block corresponding to the window.

（4）Press  to confirm the selection and to return the previous menu, press to cancel the selection operation and the system return the previous menu.

3. Search parameters, commands and network

（1）Search the block windows of the required commands, the parameters and the network ,i.e. press , ,  ,  to select the windows, and the ladder program of the corresponding block are displayed to search the commands, the parameters, the network and so on.

（2）Press to enter the search window shown in Fig. 3-24:

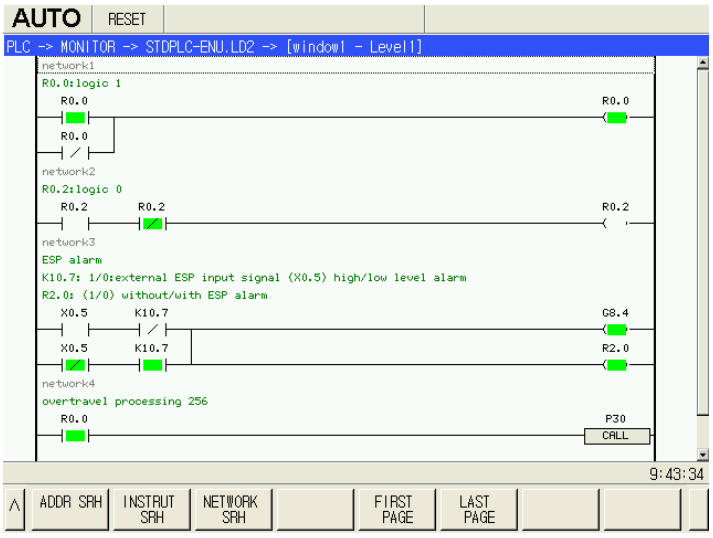


Fig. 3-24

（3）Separately press , to search the corresponding parameters, commands and network in the blocks corresponding to the windows, and the cursor positions to the

corresponding to the corresponding positions.

（4）Press FIRST PAGE, LAST PAGE to position the HOME and the END of the blocks of the corresponding blocks.

（5）Press ^, and the system returns the previous menu.

4. Return

In the figure, press ^ and the system returns to the previous menu.

3.3.5.2 Searching and setting PLC Value

In the ladder window, press PLC DATA to enter PLC Value state display window including K, D, DT, DC parameter setting, shown in Fig. 3-25:

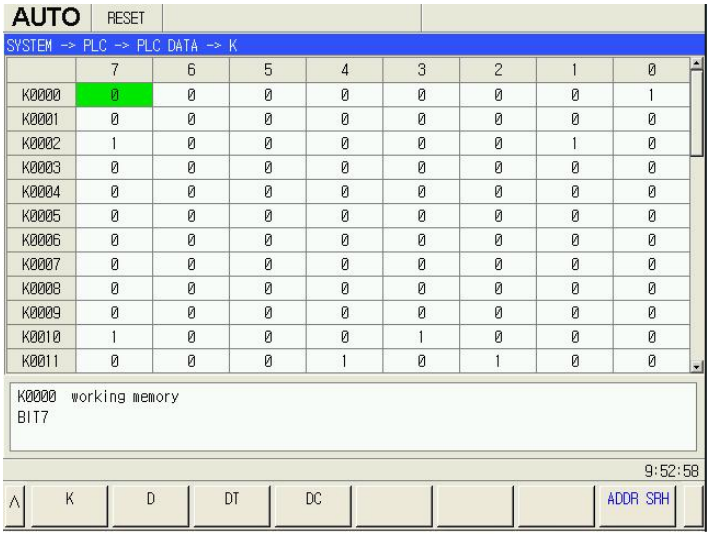


Fig. 3-25

1. K parameter setting

（1）In PLC Value state display window, press K to enter K parameter setting display window.

（2）Press , , , , , , to select the required modifying parameter state bit; or press ADDR SRH to input the selected K variable, press OK and the cursor positions to the parameter.

（3）Press repetitively INPUT to switch the state bit 0 and 1, and to modify the state of K parameter state bit.

（4）Press , , , , to move the cursor and the modification is completed.

Press ADDR SRH, input the required K parameter address to position the cursor to the K parameter address.

2. D parameter setting

（1） In PLC Value state display window, press [D] to enter D parameter setting display window shown in Fig. 3-26:

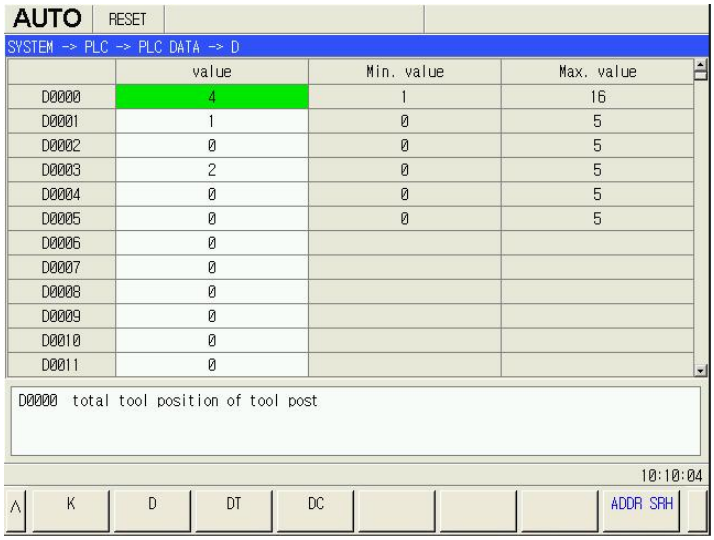


Fig. 3-26

（2） Press to select the required D parameter; or press [ADDR SRH] and input the required D parameter, press [OK] and the cursor positions to the parameter. The parameter meaning is displayed in the blew of the window;

（3）Press [INPUT], to make the selected D parameter in the state of modification.

（4）Input the modification value and press [INPUT], and the modification is completed.

3. DT parameter setting

（1）In PLC Value state display window, press [DT] to enter DT parameter setting display window shown in Fig. 3-27:

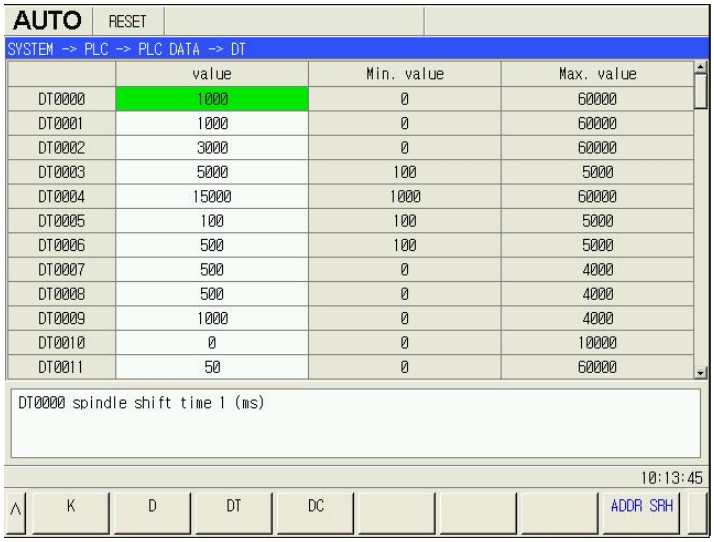


Fig. 3-27

The setting method of DT parameter is the same that of D parameter.

1. DC parameter setting

In PLC Value state display window, press DC to enter DT parameter setting display window shown in Fig. 3-28:

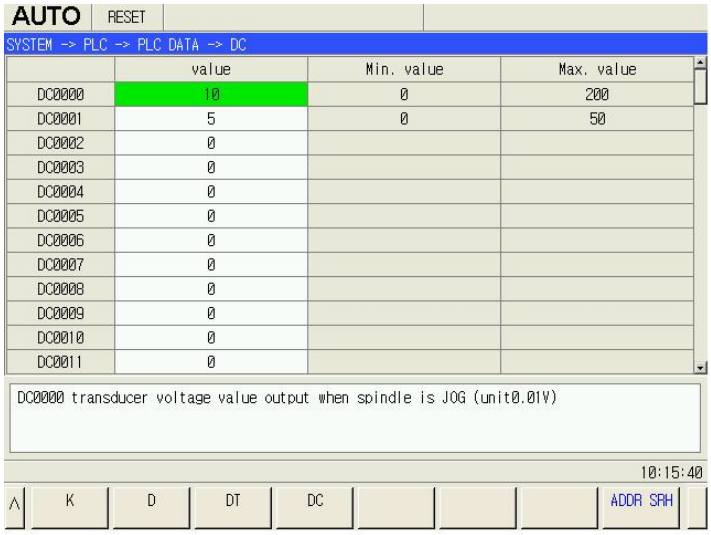


Fig. 3.-28

The setting method of DT parameter is the same that of D parameter.

3.3.5.3 PLC state search display

In the ladder window, press > and PLC STATE to enter PLC state display window shown in Fig. 3-29:

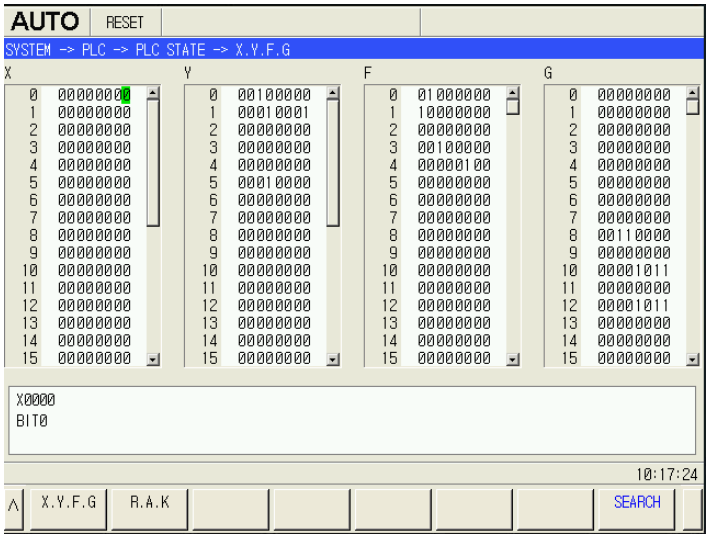


Fig. 3-29

In the window, press X.Y.F.G and the system displays the state message of X, Y, F, G. at the moment, press ⇦, ⇨ to switch X, Y, F, G, press ⇧, ⇩ to view X, Y, F, G.

There is the annotation of each parameter below the window when the parameter is viewed. Press CHANGE to view the detailed annotation of each bit of each parameter.

Press SEARCH to position the cursor to search the parameter position. The search can be done in the whole window, and the parameter name and the parameter name must be input correctly.

Press ∧ and the window display returns to the previous menu.

3.4 Setting Window

Press SETTING to access the setting window. It mainly includes windows of the tool offset and CNC setting and macro variable, etc. The content can be checked through the corresponding soft keys. The structure of the soft key layers is shown as below:

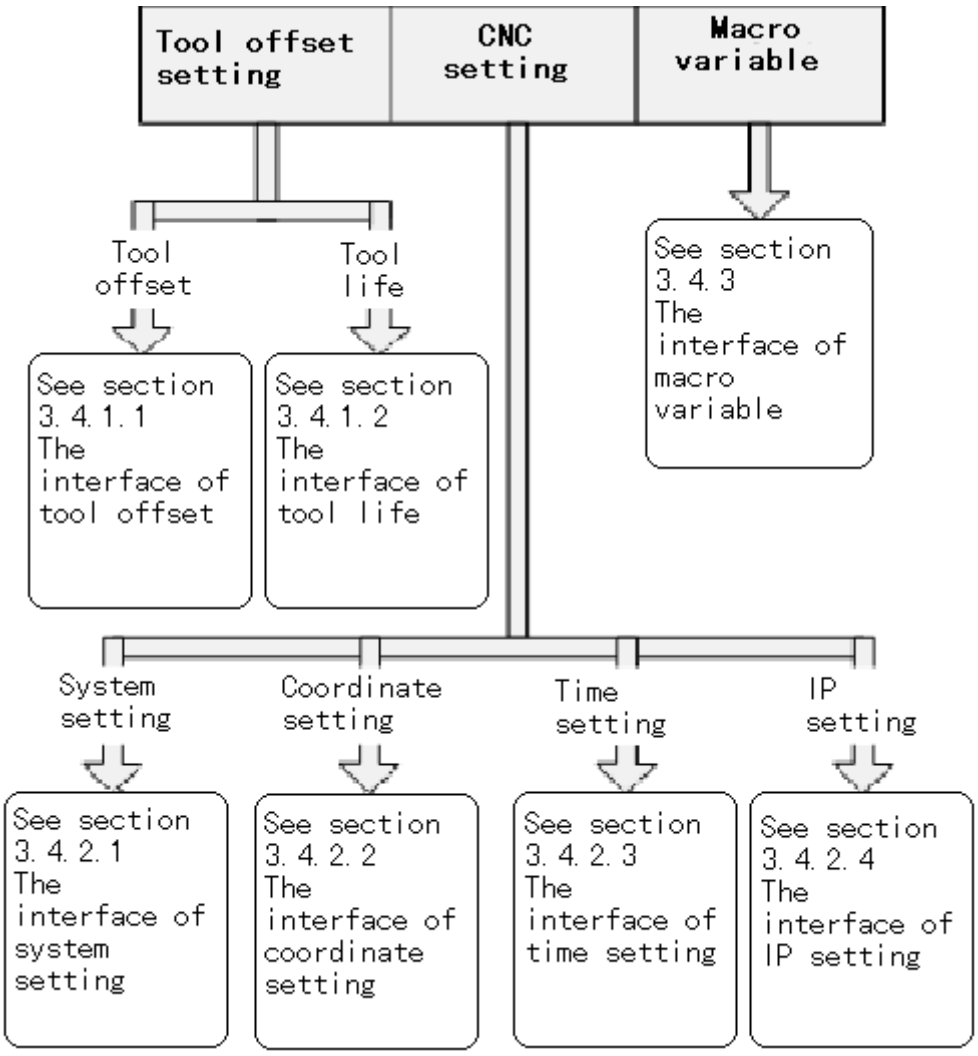

Tool offset setting
CNC setting
Macro variable
Tool offset
Tool life
See section 3.4.3 The interface of macro variable
See section 3.4.1.1 The interface of tool offset
See section 3.4.1.2 The interface of tool life
System setting
Coordinate setting
Time setting
IP setting
See section 3.4.2.1 The interface of system setting
See section 3.4.2.2 The interface of coordinate setting
See section 3.4.2.3 The interface of time setting
See section 3.4.2.4 The interface of IP setting

Fig. 3-32

3.4.1 Tool offset setting

3.4.1.1 Tool offset setting

Press TOOL OFFSET to access the tool compensation window shown in Fig.3-33:

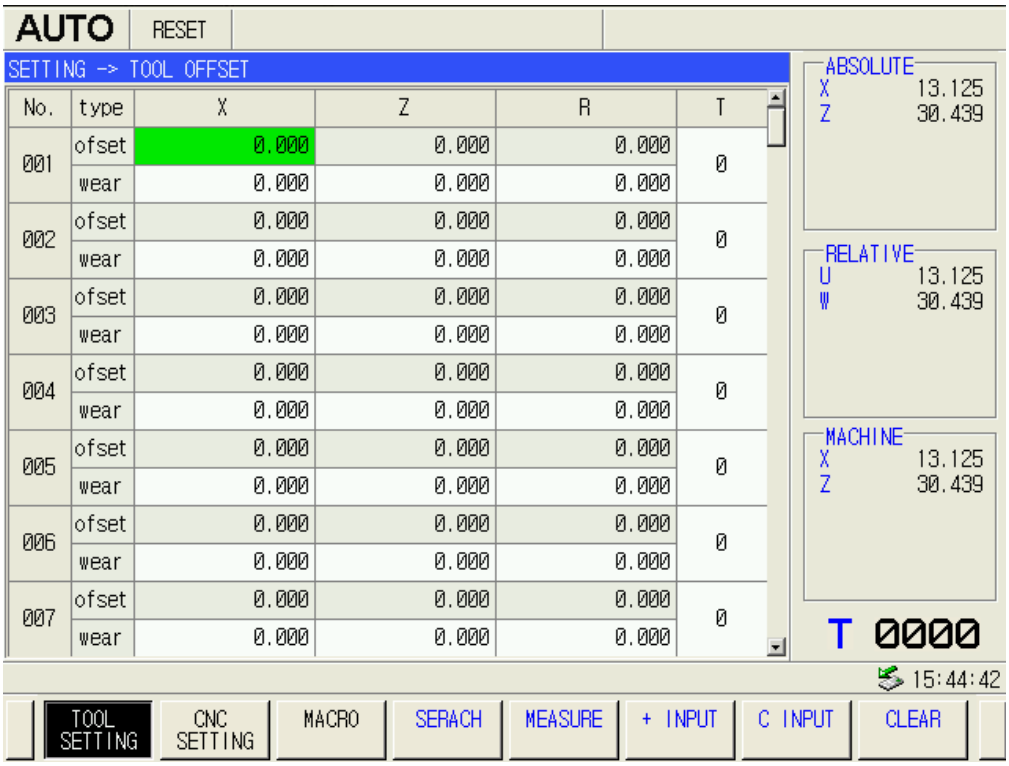


Fig.3-31

In the window, the user can search and set the too offset value and the wear value for each axis corresponding to each tool offset number, and the concrete setting methods are referred to Chapter 6.6.

In the right side column of the tool offset setting, the system simultaneously displays the current absolute coordinate values, the relative coordinate values and the tool number of the current program running.

Note 1: The displayed axes in the window are set by the parameter No.1010 and No.8130, the rotary axis is not displayed, the tool offset value is valid to the linear axis instead of the rotary axis.

Note 2: The linear axis and the rotary axis are specified (cannot be 0) by the axis attribution for each axis of No.1022.

Note 3: The name for each axis is set by No.1020.

Note 4: No. 5004 Bit1 sets the tool offset value for each axis to be the diameter or the radius value designation; No.1006 Bit3 sets the amount of movement for each axis to be the diameter or the radius designation.

Note 5: When the operation authority level is more than [4], the tool offset setting, the wear value setting and others can be executed.

Note 6: The system supports the tool offset of the most 4 linear axes; when the linear axes is more than 4, the system only displays 4 linear axes.

3.4.1.2 Tool life

Press[TOOL LIFE] to enter the tool offset setting window, which is shown in Fig.3-34:

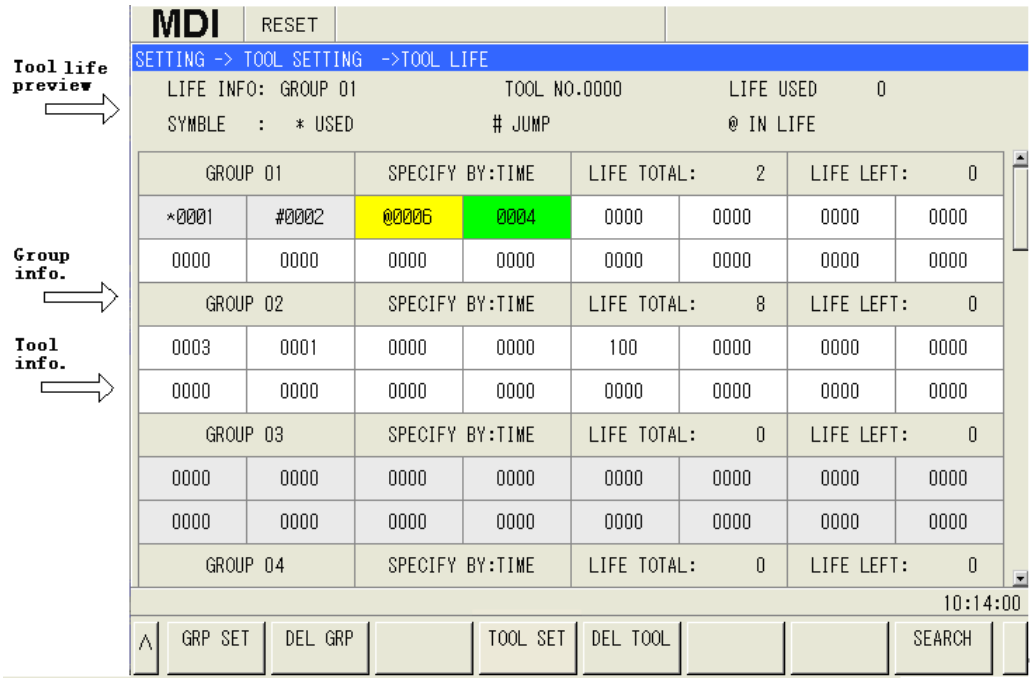

MDI
RESET
SETTING -> TOOL SETTING ->TOOL LIFE
LIFE INFO: GROUP 01 TOOL NO.0000 LIFE USED 0
SYMBLE : * USED # JUMP @ IN LIFE
GROUP 01 SPECIFY BY:TIME LIFE TOTAL: 2 LIFE LEFT: 0
*0001 #0002 00006 0004 0000 0000 0000 0000
0000 0000 0000 0000 0000 0000 0000 0000
GROUP 02 SPECIFY BY:TIME LIFE TOTAL: 8 LIFE LEFT: 0
0003 0001 0000 0000 100 0000 0000 0000
0000 0000 0000 0000 0000 0000 0000 0000
GROUP 03 SPECIFY BY:TIME LIFE TOTAL: 0 LIFE LEFT: 0
0000 0000 0000 0000 0000 0000 0000 0000
0000 0000 0000 0000 0000 0000 0000 0000
GROUP 04 SPECIFY BY:TIME LIFE TOTAL: 0 LIFE LEFT: 0
10:14:00
GRP SET
DEL GRP
TOOL SET
DEL TOOL
SEARCH
Tool life preview
Group info.
Tool info.

Fig. 3-34

Tool explanation column: the first line display the message of the tool where the cursor is. The second displays the definition of tool message color. The user can visually definitions of tool frame with different color.

Tool group message column: the first line displays the current tool group message, including current tool group, counting method of tool group, reset value of the tool group, used life of the selected tool in the tool group.
Tool message column: display the tool message in the tool group

1. Modifying tool group data

In MDI mode, press [GROUP SETTING] to pop up a dialog box, select the counting method (time or times) of the tool group and set the life value of the tool group as Fig. 3-35:

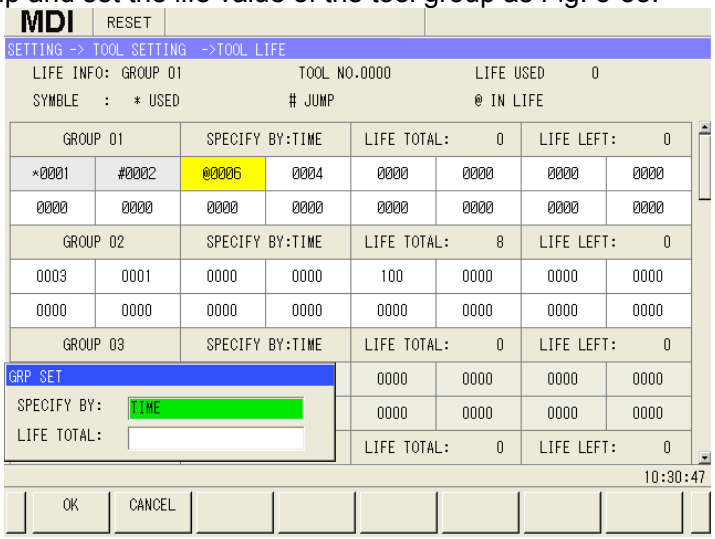


Fig. 3-35

2. Setting tool status

In MDI mode, move the cursor the tool number, press [TOOL STATUS SETTING] to set the current tool to the skip or cancel skip status as Fig. 3-36:

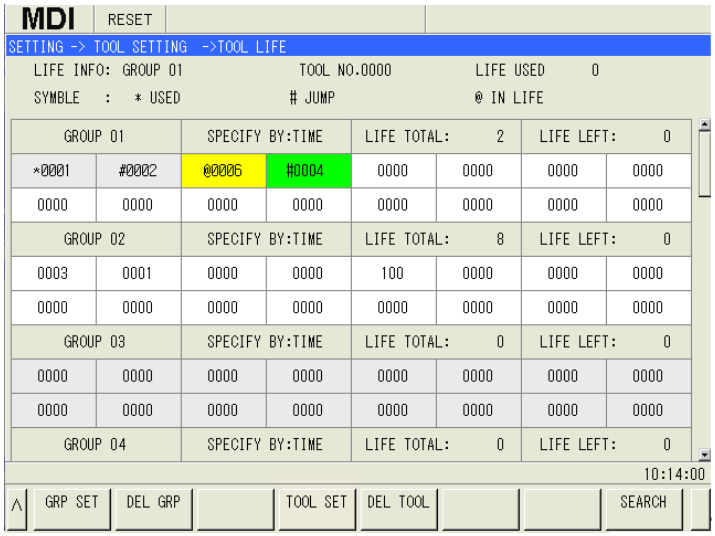

MDI
RESET
SETTING -> TOOL SETTING ->TOOL LIFE
LIFE INFO: GROUP 01 TOOL NO.0000 LIFE USED 0
SYMBLE : * USED # JUMP @ IN LIFE
GROUP 01 SPECIFY BY:TIME LIFE TOTAL: 2 LIFE LEFT: 0
*0001 #0002 @0005 #0004 0000 0000 0000 0000
0000 0000 0000 0000 0000 0000 0000 0000
GROUP 02 SPECIFY BY:TIME LIFE TOTAL: 8 LIFE LEFT: 0
0003 0001 0000 0000 100 0000 0000 0000
0000 0000 0000 0000 0000 0000 0000 0000
GROUP 03 SPECIFY BY:TIME LIFE TOTAL: 0 LIFE LEFT: 0
0000 0000 0000 0000 0000 0000 0000 0000
0000 0000 0000 0000 0000 0000 0000 0000
GROUP 04 SPECIFY BY:TIME LIFE TOTAL: 0 LIFE LEFT: 0
10:14:00
GRP SET
DEL GRP
TOOL SET
DEL TOOL
SEARCH

Fig. 3-36

3. Modifying the tool number

In MDI mode, move the cursor to the tool number, directly input the tool number to modify, simultaneously clear the tool life of current tool as Fig. 3-37:

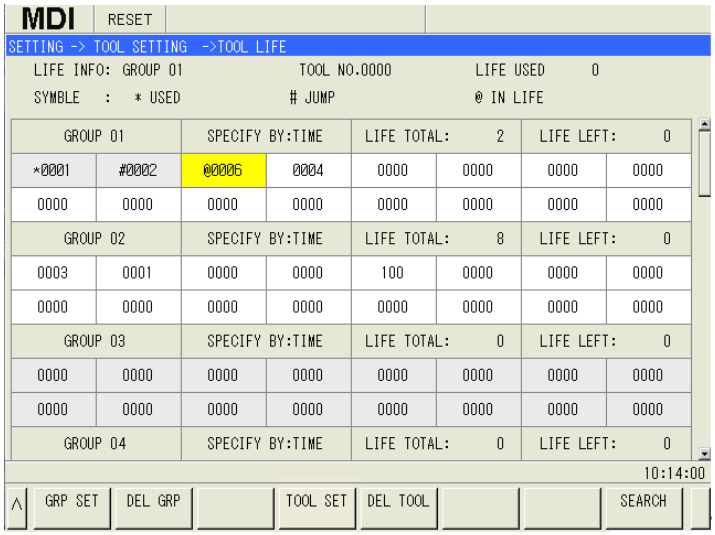

MDI
RESET
SETTING -> TOOL SETTING ->TOOL LIFE
LIFE INFO: GROUP 01 TOOL NO.0000 LIFE USED 0
SYMBLE : * USED # JUMP @ IN LIFE
GROUP 01 SPECIFY BY:TIME LIFE TOTAL: 2 LIFE LEFT: 0
*0001 #0002 @0005 0004 0000 0000 0000 0000
0000 0000 0000 0000 0000 0000 0000 0000
GROUP 02 SPECIFY BY:TIME LIFE TOTAL: 8 LIFE LEFT: 0
0003 0001 0000 0000 100 0000 0000 0000
0000 0000 0000 0000 0000 0000 0000 0000
GROUP 03 SPECIFY BY:TIME LIFE TOTAL: 0 LIFE LEFT: 0
0000 0000 0000 0000 0000 0000 0000 0000
0000 0000 0000 0000 0000 0000 0000 0000
GROUP 04 SPECIFY BY:TIME LIFE TOTAL: 0 LIFE LEFT: 0
10:14:00
GRP SET
DEL GRP
TOOL SET
DEL TOOL
SEARCH

Fig. 3-37

4. Delete the tool

In MDI mode, move the cursor to the tool number, press [DELETE TOOL], set the current tool number to zero, simultaneously clear the tool life of the tool number.

5. Delete group

In MDI mode, press [DELETE GROUP] to delete all tools in the group, i.e. clear the tool number in the group, simultaneously and clear the reset life in the group as Fig.3-38.

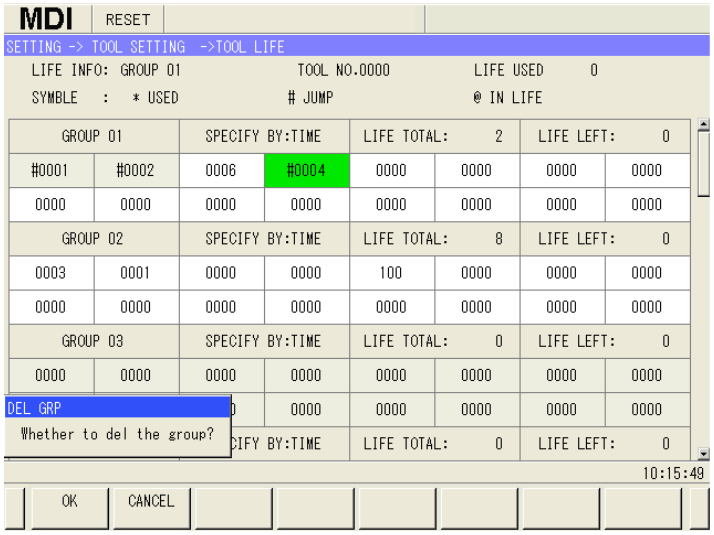

MDI
RESET
SETTING -> TOOL SETTING ->TOOL LIFE
LIFE INFO: GROUP 01 TOOL NO.0000 LIFE USED 0
SYMBLE : * USED # JUMP @ IN LIFE
GROUP 01 SPECIFY BY:TIME LIFE TOTAL: 2 LIFE LEFT: 0
#0001 #0002 0006 #0004 0000 0000 0000 0000
0000 0000 0000 0000 0000 0000 0000 0000
GROUP 02 SPECIFY BY:TIME LIFE TOTAL: 8 LIFE LEFT: 0
0003 0001 0000 0000 100 0000 0000 0000
0000 0000 0000 0000 0000 0000 0000 0000
GROUP 03 SPECIFY BY:TIME LIFE TOTAL: 0 LIFE LEFT: 0
0000 0000 0000 0000 0000 0000 0000 0000
DEL GRP
Whether to del the group?
10:15:49
OK
CANCEL

Fig. 3-38

6. Searching the tool

Press SEARCH button, input the tool number to search the tool as Fig. 3-39.

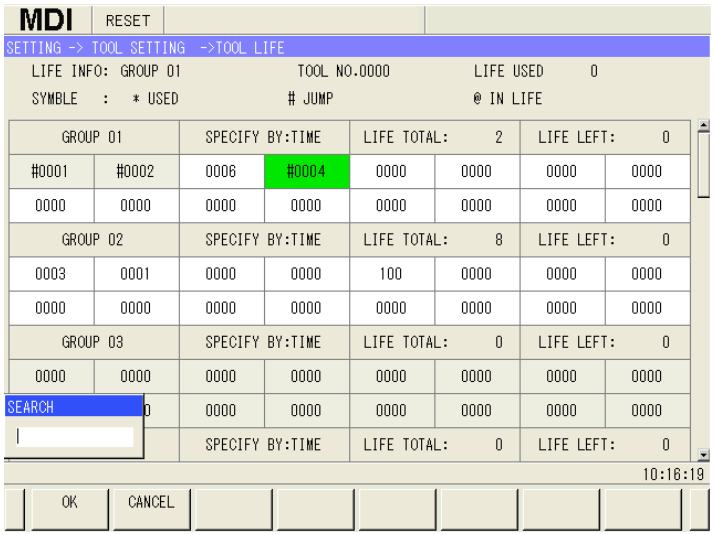
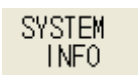

MDI
RESET
SETTING -> TOOL SETTING ->TOOL LIFE
LIFE INFO: GROUP 01 TOOL NO.0000 LIFE USED 0
SYMBLE : * USED # JUMP @ IN LIFE
GROUP 01 SPECIFY BY:TIME LIFE TOTAL: 2 LIFE LEFT: 0
#0001 #0002 0006 #0004 0000 0000 0000 0000
0000 0000 0000 0000 0000 0000 0000 0000
GROUP 02 SPECIFY BY:TIME LIFE TOTAL: 8 LIFE LEFT: 0
0003 0001 0000 0000 100 0000 0000 0000
0000 0000 0000 0000 0000 0000 0000 0000
GROUP 03 SPECIFY BY:TIME LIFE TOTAL: 0 LIFE LEFT: 0
0000 0000 0000 0000 0000 0000 0000 0000
SEARCH
10:16:19
OK
CANCEL

Fig. 3-39

Note 1: The quantity of tool group is determined by No.6813 and No6800#0. When No.6813 value is less than 8, the default is 128 groups.

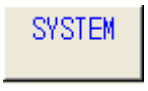
Note 2: The tool group which life and counting method have been set cannot beset again, and is done again after it is deleted.

3.4.2 CNC setting window

On the setting window, press 

to access CNC system setting window and it mainly includes the system and the coordinate setting, system time and IP.

3.4.2.1 System setting window

On CNC setting window, press 

to access the system setting window, which sets the program and the parameter switches, auto sequence number and input units, etc, which is shown as

below:

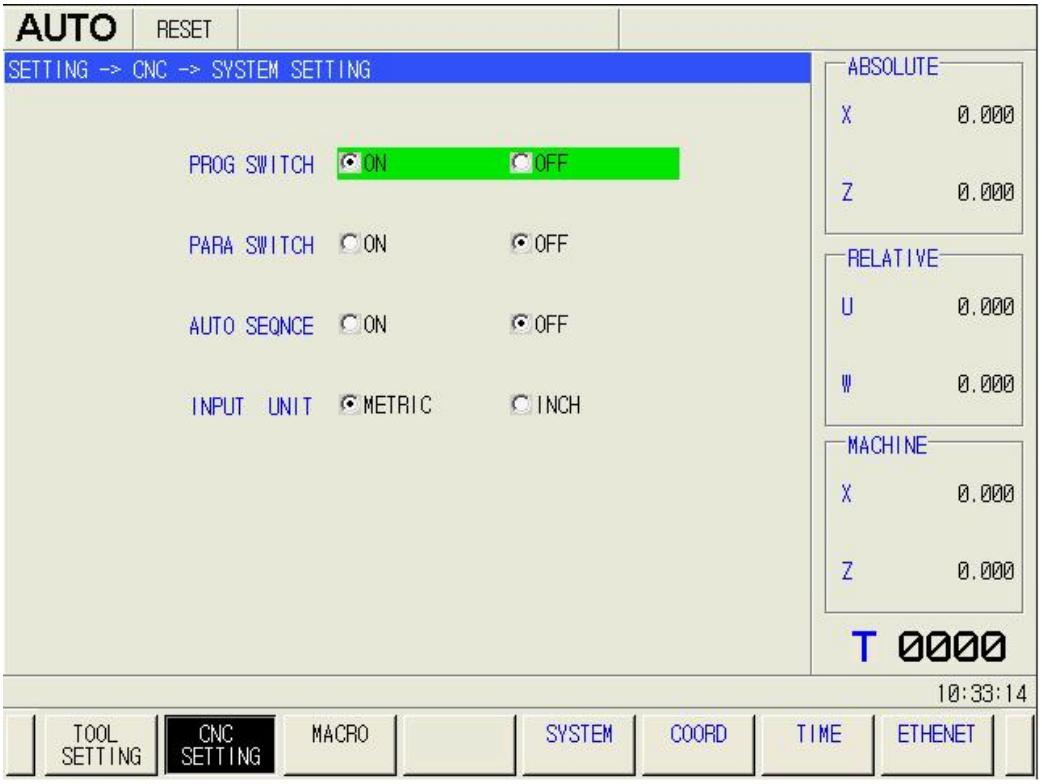


Fig.3-40

On the window, it mainly sets on or off of the program and the parameter switches, etc.

On the window, press ⇧ or ⇩ to switch among the program switch, the parameter switch, auto sequence number and the input units, etc; In MDI, when the operation authority is above level (3), press ⇦, ⇨ or INPUT to select on/off and metric /inch system.

In the right column, it also displays the current absolute position coordinate and the relative coordinate position value and the tool number of current running program.

Note 1: Only when the program switch or the parameter switch is on, can the program and the parameter be edited, rewritten or set.

Note 2: Only when the operation authority level is more than [3], can CNC system be set.

Note 3: Only when the program protection switch is on, which is installed on the machine panel, can the program and the parameter switches on/off be set.

3.4.2.2 Coordinate setting window

Press SETTING to access the setting window; on CNC setting window, press COORD to access the coordinate setting window, which is shown as below:

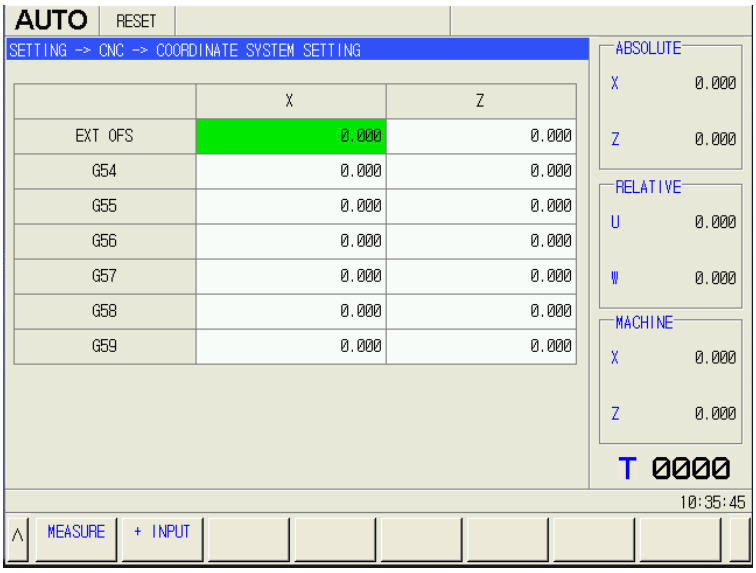


Fig.3-41

In coordinate setting window, it displays the origin offset value for each axis and the offset value of each coordinate axis in each coordinate system. Set the origin offset value relative to each axis and the offset value of each coordinate axis in each coordinate system.

In coordinate setting window, press ⇧ or ⇩ to select the coordinate system to be set, and press ⇦ or ⇨ to select the coordinate axis to set the offset; there are three kinds of modifying offset value: direct input, measure input and + input;

Direct input: select the coordinate axis to be modified, press INPUT to input the offset value, and the operation is completed.

Measure input: select the coordinate system to be modified, press MEASURE, input the measured value(for X axis, input X--, for Z axis, input Z---), and so the operation is completed.

+ input: it is used to modify the input offset value and is the incremental input. For example: X adds the offset value -0.2mm in G54 coordinate system, the cursor moves to X position in G54 coordinate system, + INPUT is pressed, -0.2 is input, and so the operation is completed.

In the right column, the system simultaneously displays the current absolute coordinate value and the relative coordinate value, and the used tool number in the current program running.

Note 1: Only in MDI mode, when the operation authority level is more than [4], the coordinate offset can be set or rewritten.

Note 2: The quantity of axes is set by parameters #1010 and #8130.

Note 3: The name for each axis is set by parameter #1020.

Note 4: The origin offset value of each coordinate in each coordinate system can be set by the parameter and the corresponding relation is shown as below:

Parameter #1220: The external work piece origin offset value for each axis.

Parameter #1221: Each axis origin offset value of work piece coordinate system 1 (G54).

Parameter #1222: Each axis origin offset value of work piece coordinate system 2 (G55).
Parameter #1223: Each axis origin offset value of work piece coordinate system 3 (G56).
Parameter #1224: Each axis origin offset value of work piece coordinate system 4 (G57).
Parameter #1225: Each axis origin offset value of work piece coordinate system 5 (G58).
Parameter #1226: Each axis origin offset value of work piece coordinate system 6 (G59).

3.4.2.3 Setting system time window

Press SETTING to access setting window; in CNC setting window, press TIME to access setting system time window, which is shown as below:

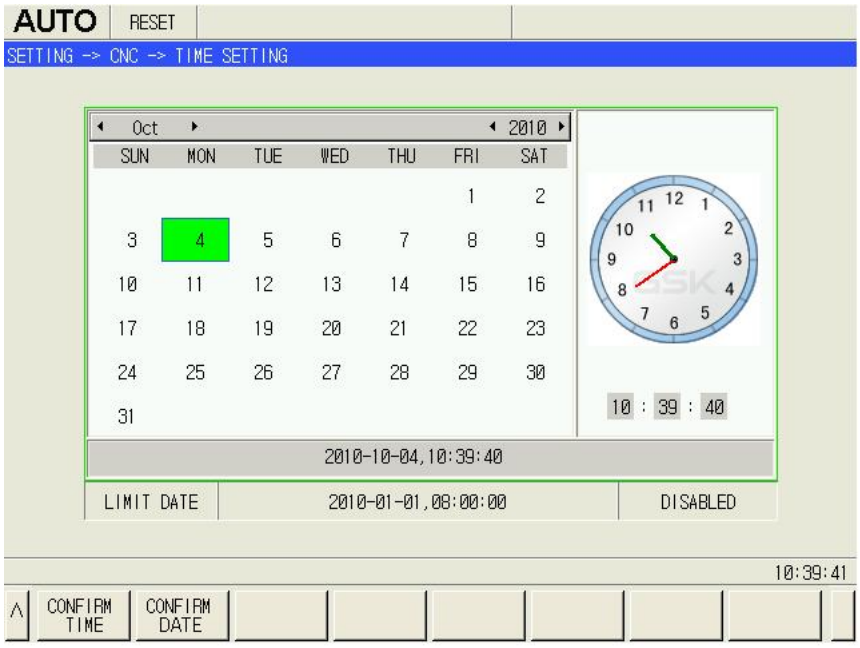


Fig.3-42

On the time setting page, press CHANGE to switch among the date, month, year and time boxes in cycle.

Setting month: press CHANGE to switch into the month box, and it changes into green, press ⇧, ⇩, ⇦ and ⇨ to change the month, press CHANGE to switch into the other box and the month setting completes.

Setting year: press CHANGE to switch into the year box and it changes into green, press ⇧, ⇩, ⇦ and ⇨ to change the year, press CHANGE to switch into the other box and the year setting completes.

Setting time: press CHANGE to switch into the time box and it changes into green, press ⇧, ⇩, ⇦ or ⇨ to select the time, press CONFIRM TIME to complete the time setting.

Stop serial number input: when the system sets the stop function in the limited time, ENABLE TRYOUT is pressed to input the releasing code to release the stop run.

3.4.2.4 Setting system IP window

Press SETTING to access the setting window; on CNC setting window, press ETHENET to access system IP setting window, which is shown as below:

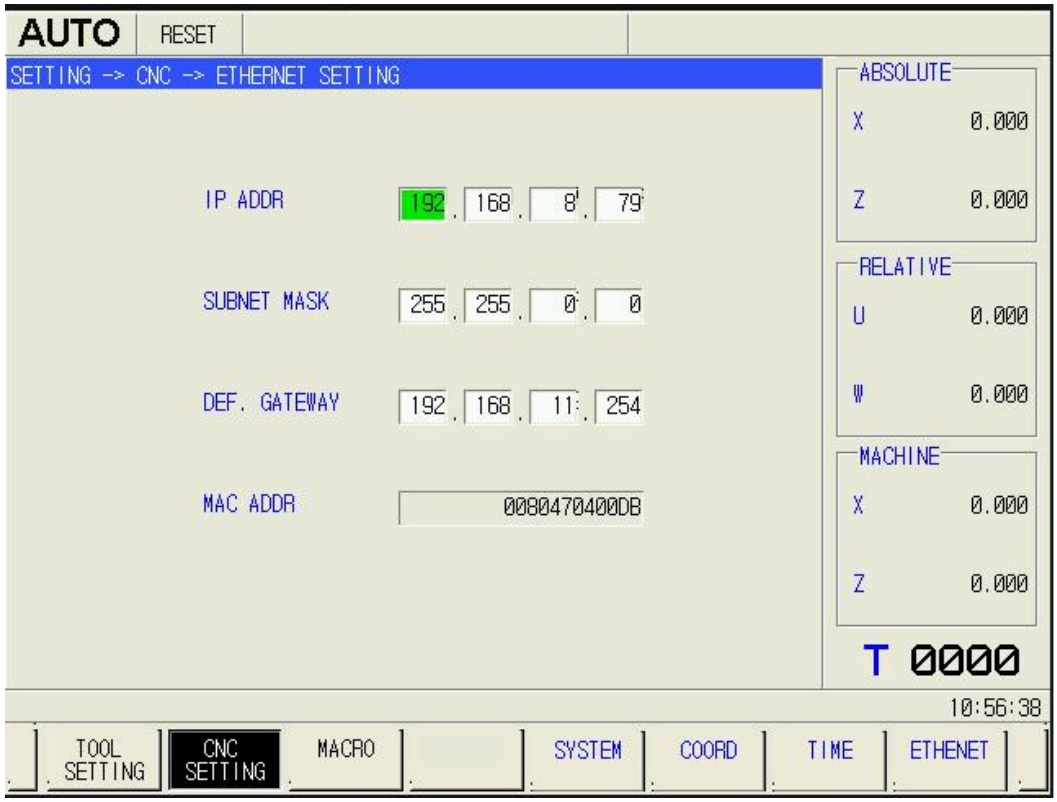


Fig.3-43

1）Press ⇧ and ⇩ to switch among IP address, subnet mast or default gateway column.

2）Press ⇦ and ⇨ to switch between each address box, input the address to be set.

3.4.2.5 Machine soft panel

To conveniently operate the system without the machine operation panel for the user, GSK988T system provides the machine soft panel. Press SETTING to enter the setting interface; in CNC setting page, press VIRTUAL PANEL the machine soft panel as Fig. 3-44:

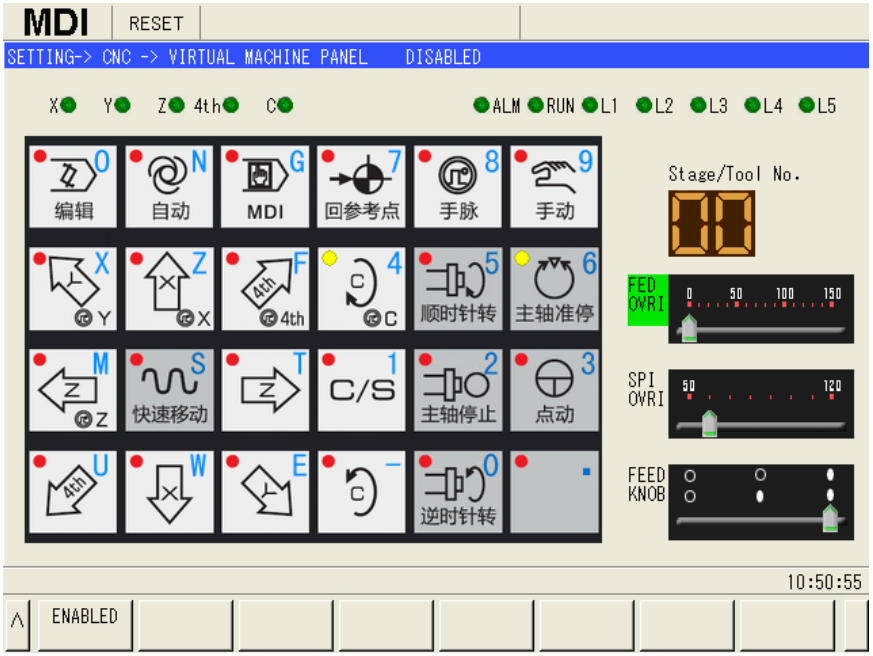


Fig. 3-44

In the page, ENABLED (see it for the user with more than 2-level authority) can be pressed, i.e. the machine soft panel can be used. The characters lablelled on the machine soft panel corresponds to the characters on the system keyboard, and the machine can be operated according to the corresponding character. Pressing , can switch to other page with other function keys. Pressing , can select the spindle strobe, spindle override and feedrate override, which can be regulated by pressing , .

Note 1: K12.7=1, i.e. the operation panel is MPUO2B, the spindle override regulation on the machine soft panel is valid and the external spindle override knob is invalid.

Note 2: K12.7=0, i.e. the operation panel is MPUO2A, the spindle override is controlled by numerical key or symbol key on the machine soft panel(there are two pages on the machine soft panel and they are converted by pressing Page Up/Down.

Note 3: When the displayed feedrate override is opposite to the actual, K10.0 is modified to get the correct.

Note 4: When the feedrate override on the machine soft panel is valid, the external feed knob is invalid.

Note 5: CYCLE START key on te machine soft panel, CYCLE START key on the machine panel and the external CYCLE START key are valid simultaneously.

3.4.3 Macro variable window

On the setting window, press MACRO to access the macro variable window, which is shown as below:

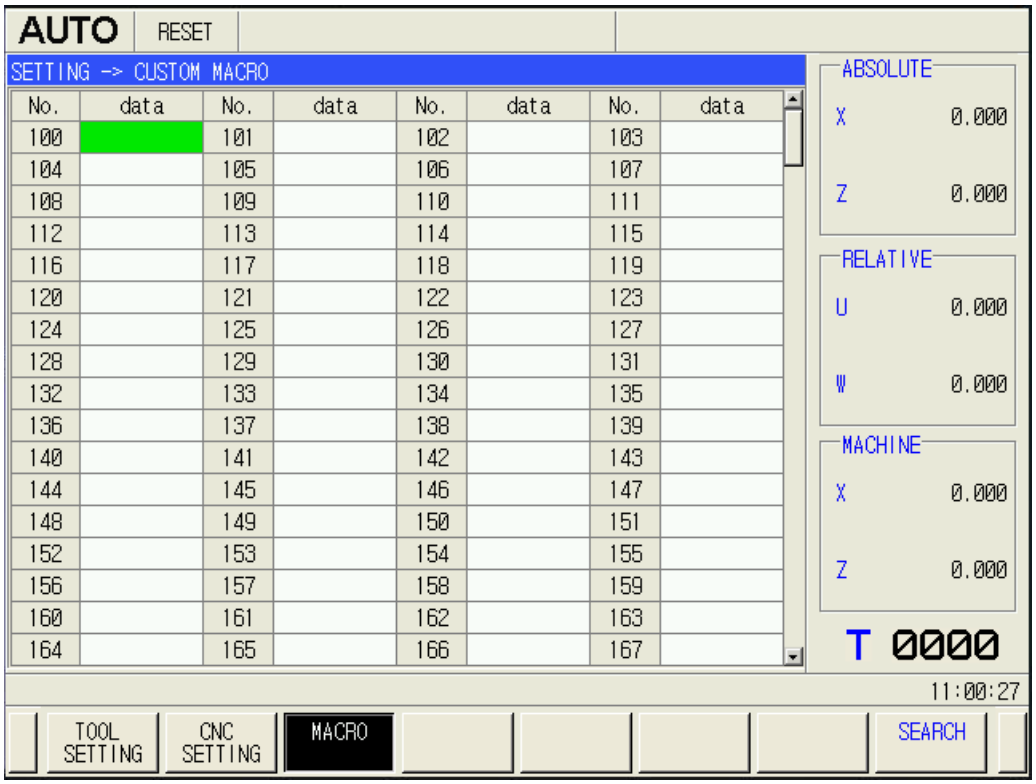
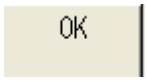


Fig.3-45

On the macro variable window, check and set the value relative to each macro variable.

On macro variable window, press , and , , , to select the macro variable to be rewritten, the selected macro variable changes into the green-based color, or press to input the macro variable serial number to be selected, and then press  and the cursor positions in the Value of the macro variable.

In MDI mode, the operation authority level is more than [4], rewrite the macro variable Value through numerical and backspace keys; or press and the macro variable Value can be rewritten, such as the macro variable Value # 100, and rewrite the macro variable Value through numerical and backspace keys.

And press , again to complete the rewriting.

3.5 Message Window

Press to access the alarm window, there are three windows of alarm message, records and diagnosis, and check the content in each window through pressing the corresponding soft keys. The structure of the software layers is shown as below:

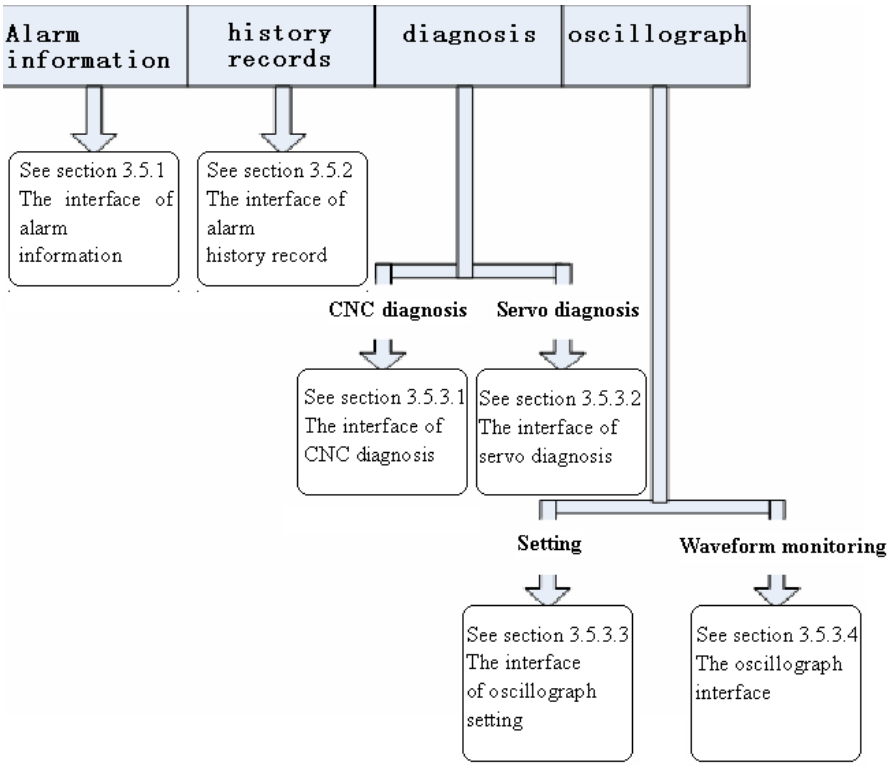

Alarm information
history records
diagnosis
oscillograph
See section 3.5.1 The interface of alarm information
See section 3.5.2 The interface of alarm history record
CNC diagnosis
Servo diagnosis
See section 3.5.3.1 The interface of CNC diagnosis
See section 3.5.3.2 The interface of servo diagnosis
Setting
Waveform monitoring
See section 3.5.3.3 The interface of oscillograph setting
See section 3.5.3.4 The oscillograph interface

Fig.3-46

3.5.1 Alarm message check window

On the message window, press ALARM MESSAGE to access the alarm message window, display the quantity CNC and PLC alarms and detailed message. The window is shown in Fig.3-38:

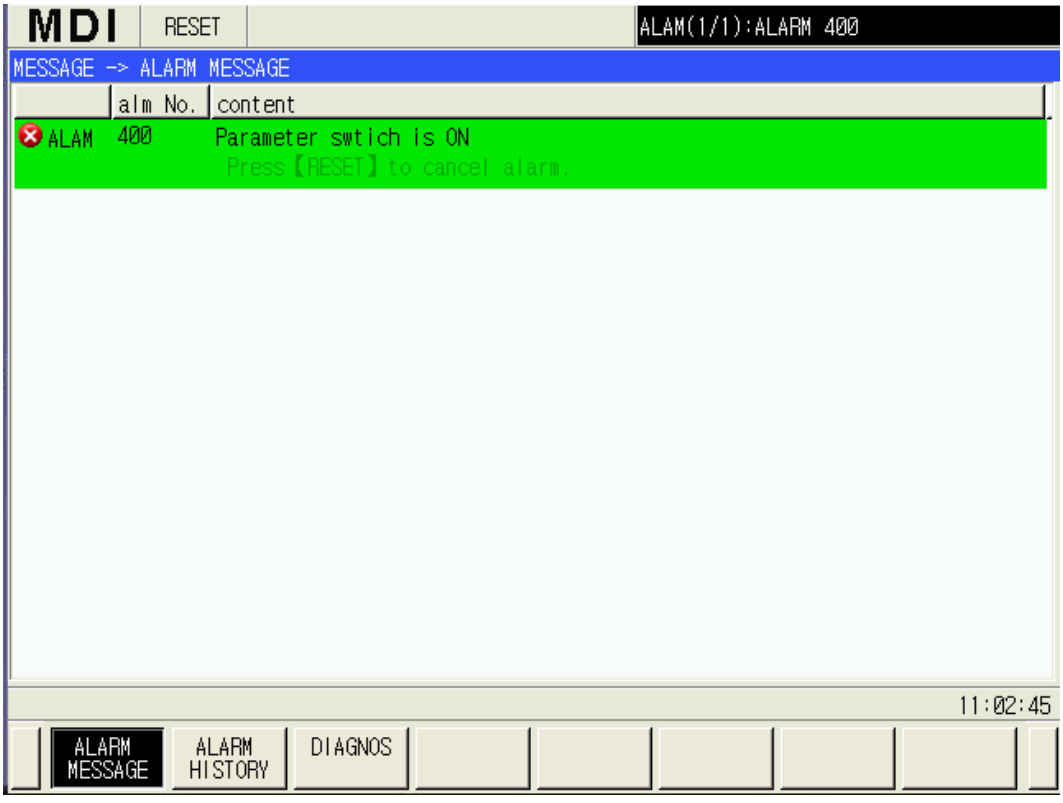


Fig.3-47

On alarm message window, the alarm message of CNC and PLC is listed in one window, and

differed through the alarm number. Press and to scroll the list line by line, or press and to scroll the list page by page.

When PLC alarms or prompts, display message of address A in black; When CNC alarms or reminds, the reasons and trouble shooting is shown as black below the message line.

Cancel alarm: Press

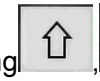
to cancel all alarms.

Note 1: When PLC alarms or reminds, the message of address A displays in green below the message line;
Note 2: When CNC alarms or reminds, the reason and the trouble shooting display in green below the message line.
Note 3: Alarms of #0——1000 are CNC, alarms of #1000——2000 are PLC, after #2000, it is prompt message.
Note 4: After the parameter is rewritten, which becomes valid after power on, the alarm can be cleared after power on again.
Note 5: The detailed alarm message and PLC alarm are referred to *Appendix I Alarm Message List and Appendix II PLC Alarm*.

3.5.2 Alarm record check window

Press

to access the message window, and then press

to access the record window. The latest alarm message is recorded on the window, including the alarm date, time, number and content. And check the alarm message through pressing  , , and . The window is shown as below:

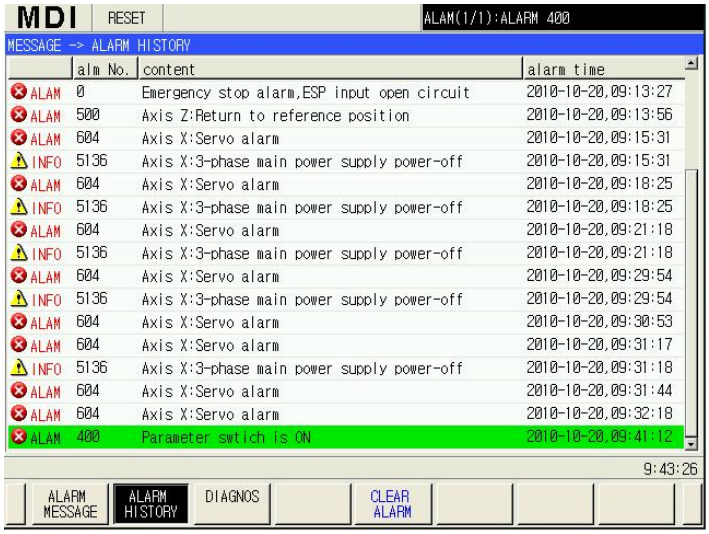

MDI
RESET
ALAM(1/1):ALARM 400
MESSAGE -> ALARM HISTORY
alm No.
content
alarm time
ALAM 0 Emergency stop alarm,ESP input open circuit 2010-10-20,09:13:27
ALAM 500 Axis Z:Return to reference position 2010-10-20,09:13:56
ALAM 604 Axis X:Servo alarm 2010-10-20,09:15:31
INFO 5136 Axis X:3-phase main power supply power-off 2010-10-20,09:15:31
ALAM 604 Axis X:Servo alarm 2010-10-20,09:18:25
INFO 5136 Axis X:3-phase main power supply power-off 2010-10-20,09:18:25
ALAM 604 Axis X:Servo alarm 2010-10-20,09:21:18
INFO 5136 Axis X:3-phase main power supply power-off 2010-10-20,09:21:18
ALAM 604 Axis X:Servo alarm 2010-10-20,09:29:54
INFO 5136 Axis X:3-phase main power supply power-off 2010-10-20,09:29:54
ALAM 604 Axis X:Servo alarm 2010-10-20,09:30:53
ALAM 604 Axis X:Servo alarm 2010-10-20,09:31:17
INFO 5136 Axis X:3-phase main power supply power-off 2010-10-20,09:31:18
ALAM 604 Axis X:Servo alarm 2010-10-20,09:31:44
ALAM 604 Axis X:Servo alarm 2010-10-20,09:32:18
ALAM 400 Parameter swtich is ON 2010-10-20,09:41:12
9:43:26
ALARM MESSAGE
ALARM HISTORY
DIAGNOS
CLEAR ALARM

Fig.3-48

Clear the alarm record: On the record window, press

to clear all records of alarms and remind message, and the window is blank after clearing.

Note: Whether clear the alarm record, which is set by parameter 3110.2.

3.5.3 Diagnosis window

Press MESSAGE to access the message window, press DIAGNOSIS to access the diagnosis window. The window is shown as below:

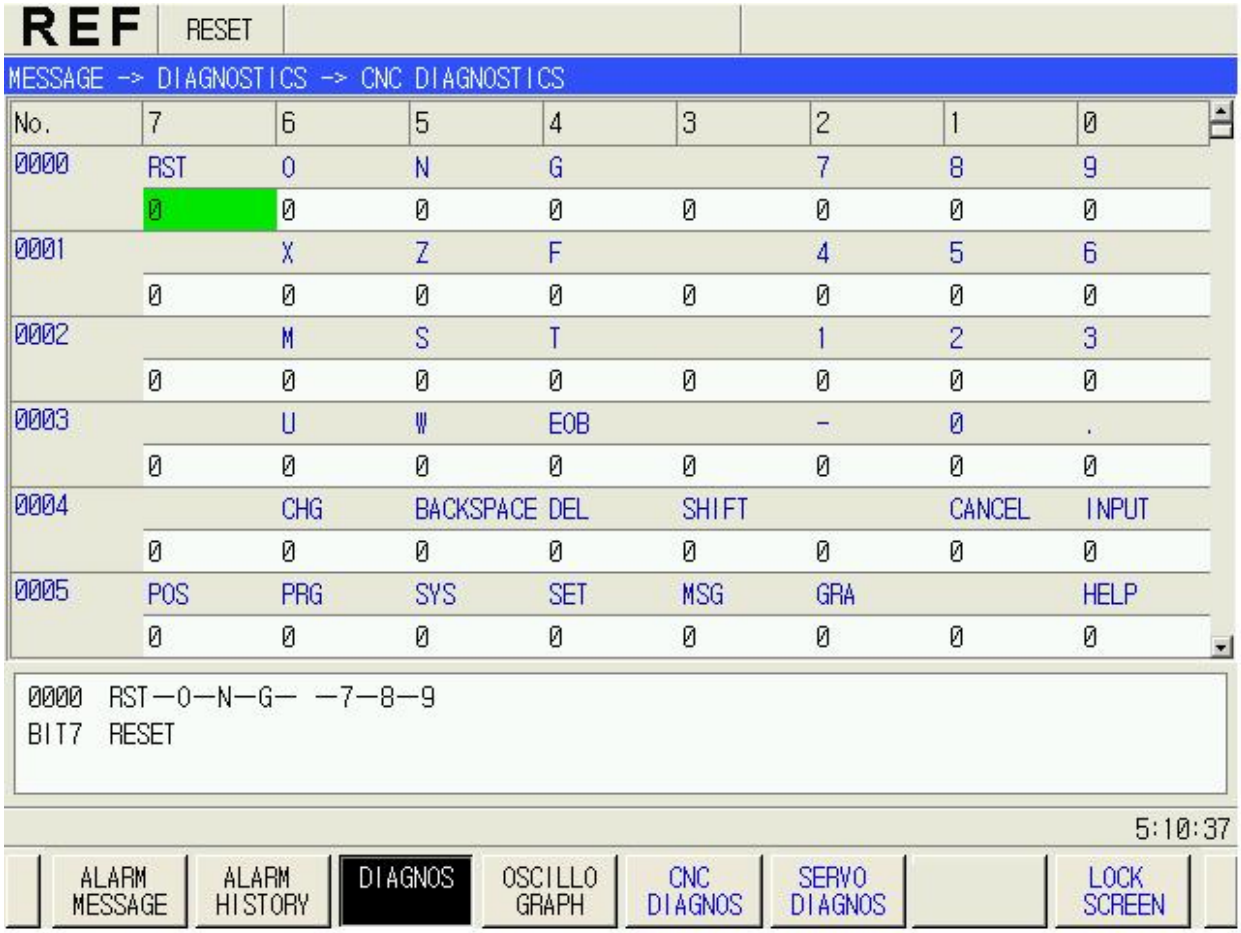

REF
RESET
MESSAGE -> DIAGNOSTICS -> CNC DIAGNOSTICS
No. 7 6 5 4 3 2 1 0
0000 RST 0 N G 7 8 9
0001 X Z F 4 5 6
0002 M S T 1 2 3
0003 U W EOB - 0 .
0004 CHG BACKSPACE DEL SHIFT CANCEL INPUT
0005 POS PRG SYS SET MSG GRA HELP
0000 RST—0—N—G— —7—8—9
BIT7 RESET
5:10:37
ALARM MESSAGE
ALARM HISTORY
DIAGNOS
OSCILLO GRAPH
CNC DIAGNOS
SERVO DIAGNOS
LOCK SCREEN

Fig.3-49

3.5.3.1 System diagnosis window

Firstly access CNC diagnosis window, press CNC DIAGNOS to access the system diagnosis window. In CNC system diagnosis window, there is message of keypad and state diagnosis and miscellaneous function parameter, etc. Press , , , , , and to check the content. To prevent the corresponding function is operated during checking some keys, such as the direction and the window keys, therefore, lock the current screen through pressing LOCK SCREEN.

On CNC diagnosis window, there are two lines to display the detailed content of the diagnosis numbers at the bottom, and the first line displays the diagnosis number; the second displays the meaning of some bit of the diagnosis number which the cursor is.

The system diagnosis window includes the diagnosis message and its corresponding diagnosis number below:

- Press key diagnosis messages on the system keyboard（diagnosis number:0~7）

The system can diagnose all keys on the system keyboard, and each key has two states of press-down and jump, they can judge whether the press key is damaged.

- Feed axis diagnosis messages(diagnosis number: 10~13）

The diagnosis number 10~13 is the diagnosis message of the servo axis 1~5. the diagnosis message of each servo axis includes input/output status of servo drive unit connected with the feed axis, pulse quantity from feed axis to FPGA, pulse quantity from FPGA to servo drive unit and the accumulated errors of feed axis's pulse (difference value between FPGA receiving's quantity and sending's quantity), and the system judges whether the feed axis works normally according to the diagnosis message.

Note：The system only displays diagnosis messages of used servo axes, does not display those of unused servo axes.

- Pulse encode diagnosis messages(diagnosis number: 30~33）

They include the rotary direction of two-channel pulse encode, Z signal state, A-, B-phase signal states and the current count pulse value, and they can judge whether the encode works normally.

- MPG diagnosis messages(diagnosis number: 40~43）

They include the rotary direction of two-channel MPG, A-, B-phase signal states and the current count pulse value, and they can judge whether the encoder works normally.

- Spindle's diagnosis messages(diagnosis number: 50~52）

They include the alarm signal, the tapping signal, enabling signal, ready signal and others of two-channel spindle.

- Diagnosis messages of machine panel（diagnosis number:60~62）

They include the accumulated error quantity, the currently continuous error quantity, and the reset quantity of machine panel, and they can judge whether the machine panel works normally.

- Diagnosis message of edit keyboard（diagnosis number：63~65）

They include the accumulated error times of edit keyboard, continuous error times of machine panel and reset times of machine panel, and they can judge whether the current machine panel works normally.

3.5.3.2 Servo diagnosis window

On the system window, press DIAGNOSIS to access the diagnosis window and press LOCK SCREEN to access the servo diagnosis window shown in Fig.3-41:

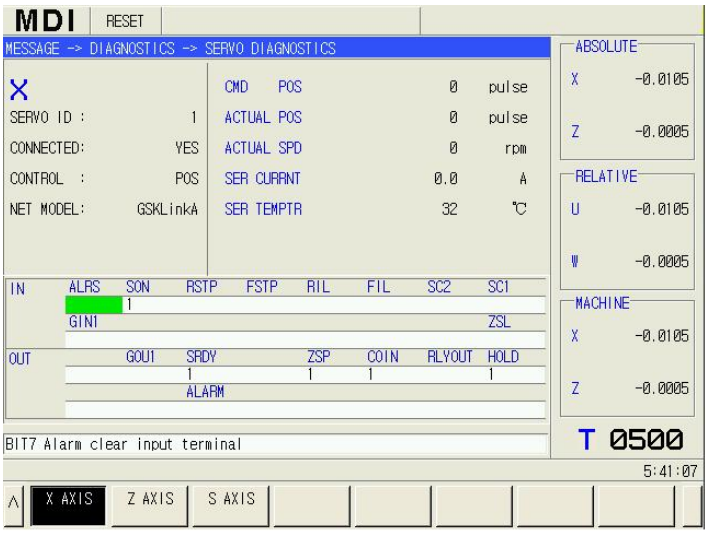


Fig.3-41

There provides the following functions in 988T servo diagnosis mode:

Through the Value of the servo communication feedback, real-time monitor the system control axis, then, the operator can learn the servo and the motor working state, etc, including:

（1）The follow error analysis of the axis, the Value are composed of two parts: The command Value received by the servo and the Value feed back by the encoder.

（2）The axial state diagnosis message: The present operating current of the servo, the motor real-time speed, the internal temperature of the servo, the servo IO point state.

（3）The servo alarm message.

The introduction of each Value in the servo diagnosis window:

X: The name of the current selected axis

Slave number: Number of the slave connecting with the axis

Connecting state: Check whether the servo communication link layers are connected.

Control mode: The diagnosis Value relative to the servo control mode, it may display as “position” and “speed”.

Command position: The quantity of the position pulses which the diagnosis Value servo receives from the system.

Feedback position: The quantity of the position pulses (not include the servo gear ratio) feed back by the diagnosis Value servo.

Command speed: The speed command value which the diagnosis Value servo receives from the system.

Motor speed: The actual speed of the diagnosis Value motor.

Spindle speed: The actual speed of the diagnosis Value spindle.

Encoder value: The current value of the diagnosis Value spindle encoder.

Servo current: Diagnosis the present operating current value of the diagnosis Value servo.

Servo temperature: The measured temperature value in the diagnosis Value servo.

IN: Value of the servo input point.

OUT: Value of the servo output point.

BIT7 alarm clear input terminal: The detailed explanation of the marked servo input and output points.

Axis switch: press X AXIS, Z AXIS, S AXIS to switch the parameters of the displayed

servo of X, Z, S axis.

Note: The servo diagnosis can display normally only when the system's servo communication function is valid, each servo system is connected correctly and the allocation of the servo slave number is correct.

3.5.4 Oscillograph window

Press MESSAGE to enter the message window and press to enter the oscillograph window. Before the oscillograph is used, the user must set the monitored servo Value, the oscillograph monitor type, the wave zoom unit and the triggered sampling time. The oscillograph setting window is shown in Fig. 3-42:

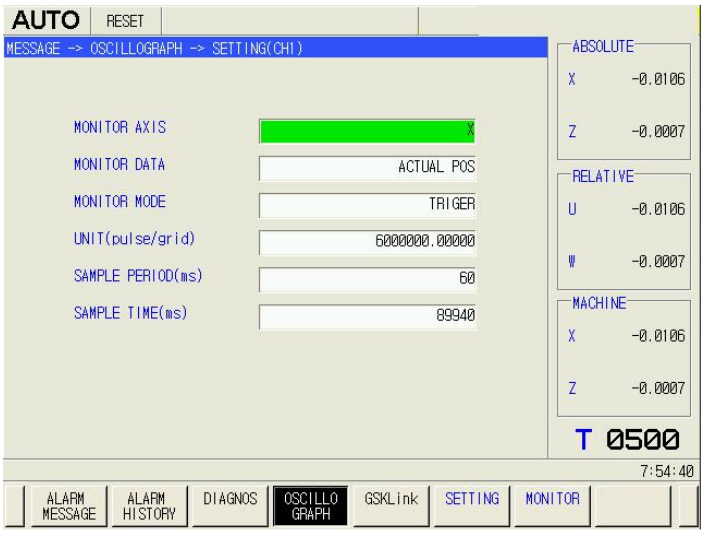


Fig.3-42

The detailed explanation of the setting content is shown below:

（1） "CH1","CH2":select the communication to be set.

（2） Monitor mode: set the oscillograph to be the trigger or the memory. The trigger: it the sampling mode is that the above setting sampling realizes the arrival time stop sampling mode. The memory: the sampling mode is that the sampling is stopped after the system has checked the servo alarm.

The difference of the two monitor modes are shown below:

Attribute Type	Sampling start mode	Sample end mode	Waveform Value saved?
Trigger	Press start/stop soft key	Automatically stops after sampling time ends	No
Memory	Press start/stop soft key	Automatically stop when the servo alarms	Automatically save

（3） Sampling period. The sampling period of GSK-CAN communication function is within

60ms, and cannot be modified manually.

（4） The other related Value is set and the setting items and the setting content are shown below:

Item	Explanation	Setting step
MONITOR AXIS	Select the axis from which the Value monitored by the current waveform is	Press INPUT to open the option box, press “UP” or “DOWN” on the MDI panel, press INPUT, so the option setting is completed.
MONITOR DATA	Select the servo Value monitored by the current waveform, and the options include: （1） Command position （2） Feedback position （3） Command speed （4） Feedback speed （5） Servo temperature （6） Servo current	As above mentioned.
UNIT(pulse/grid)	Set the wave unit displayed on the vertical axis. Taking example of the command position: Setting the unit to be 5000 means that the height of each cell in the oscillograph background has 5000 pulses	After the digit is directly input, INPUT is pressed and so the modification is completed.
SAMPLE PERIOD(ms)	Set the sampling time limit of trigger oscillograph.	Cannot modify it.

After the oscillograph Value is set, MONITOR is pressed to enter the oscillograph monitor window shown in Fig. 3-43:

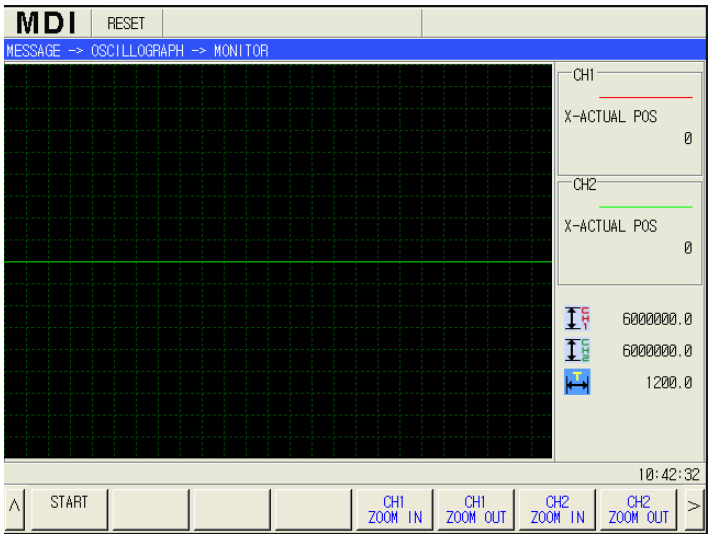


Fig. 3-43

In the oscillograph window, press “START” soft key to sample the servo Value. When the sampling Value is 0, the waveform is drawn from the center of the left edge to the right edge in the oscillograph, which path stands 0 separate position of monitor Value, the positive(+) value of the monitor Value is distributed in the upper of 0 separate position, the negative (-) value is under the 0 separate position. Such is the same as the display of the real oscillograph.

When the sampling is being executed, the soft key “STOP” is pressed to end the sampling. For the memory oscillograph, “STOP” is pressed to automatically save the sampling Value of the last 1500 pulses.

Graph introduction:

: CH1（Channel Ⅰ）wave form

: CH2（Channel Ⅱ）wave form

: CH1 Value unit

: CH2 Value unit

: Time axis unit

MONITOR -history data(sample time:2010-10-11,09:32:46): Check the historical waveform Value in the oscillograph in memory mode and the sampling time of waveform Value.

Note:

（1） When the sampling cycle is 40m, the longest time limit of the historical Value recorded by the system is about 1 min. If it exceeds the time limit, it will auto cover the previous memory area.

（2） The unit of the monitor property Value is same as that of the monitor Value corresponding to the servo diagnosis window.

Introduction of the soft key function

STOP: Button of start/stop of the control Value sample, it displays as “stop” during sampling, and it displays as “start” when the sampling stops.

CH1 ZOOM IN | CH1 ZOOM OUT | CH2 ZOOM IN | CH2 ZOOM OUT :Respectively scale the wave forms of CH1 and

CH2 in the vertical axis.

TIME ZOOM IN TIME ZOOM OUT: Scale the setting units of the time axis.

HISTORY DATA: Press it to open the history Value of Read in the historical Value during sampling in the pulses automatically saved when the last memory oscillograph stops during the memory sampling.

FINE-TUNE: The button is compound. When the history waveform is viewed during the sampling stop, “point-to-point view history Value” can view the sampling value of each sampling point along the time axis; “grid view history Value” can fasten the browse speed of history Value. The button is not display during sampling.

Besides, the oscillograph executes the double waveforms to move up, move down, move to left, move to right, page up and page down. The page up, page down are valid during the sampling stop. The above operations can be realized by the corresponding keys on the MDI panel.

Note : The oscillograph can normally display only when the system servo communication function is valid, and the servo slave allocation is correct.

3.5.5 GSK-CAN window

1）Press MESSAGE to enter the message window, press GSKLink to enter GSK-CAN window. The user can find the all servo drive unit types connected to the current system, software version, the serial number of the drive unit, the serial number of all motors and so on shown in Fig. 3-44.

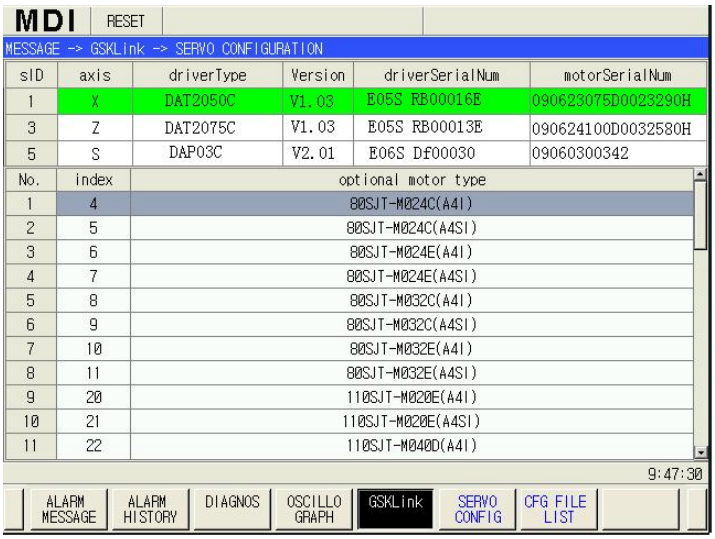


Fig. 3-44

2）Press CFG FILE LIST in GSK-CAN window to switch to the configuration file directory. The files are the ones of the drive unit shown in Fig. 3-45. Deleting the configuration file affects the system run, please do not delete it at will.

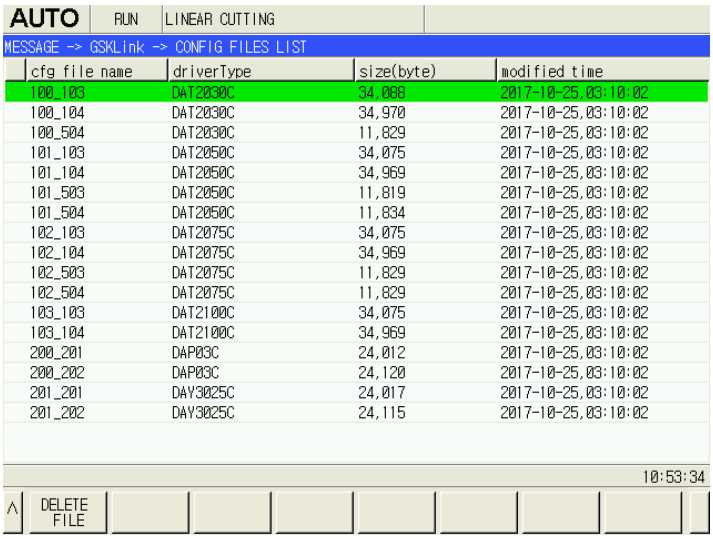


Fig. 3-45

Note : The detailed operation about GSK-CAN is referred to *GSK988T Installation and Debugging User Manual*.

3.6 Graph Window

Press GRAPH to access the graph windows, and it mainly includes the windows of the graph setting, the path display and the simulation graph, etc, and check the content of each window through pressing the corresponding soft keys. The structure of the software layers is shown in Fig.3-46:

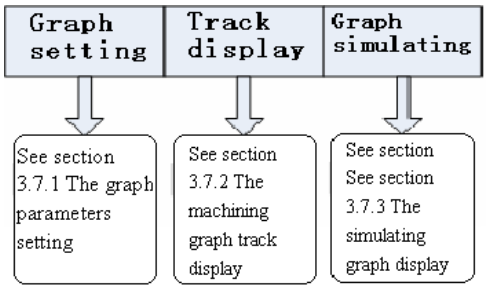

Graph setting
Track display
Graph simulating
See section 3.7.1 The graph parameters setting
See section 3.7.2 The machining graph track display
See section
See section 3.7.3 The simulating graph display

Fig.3-46

3.6.1 Setting graph parameter

On the graph window, press GRAPHSET to access the setting graph window and it is shown as below:

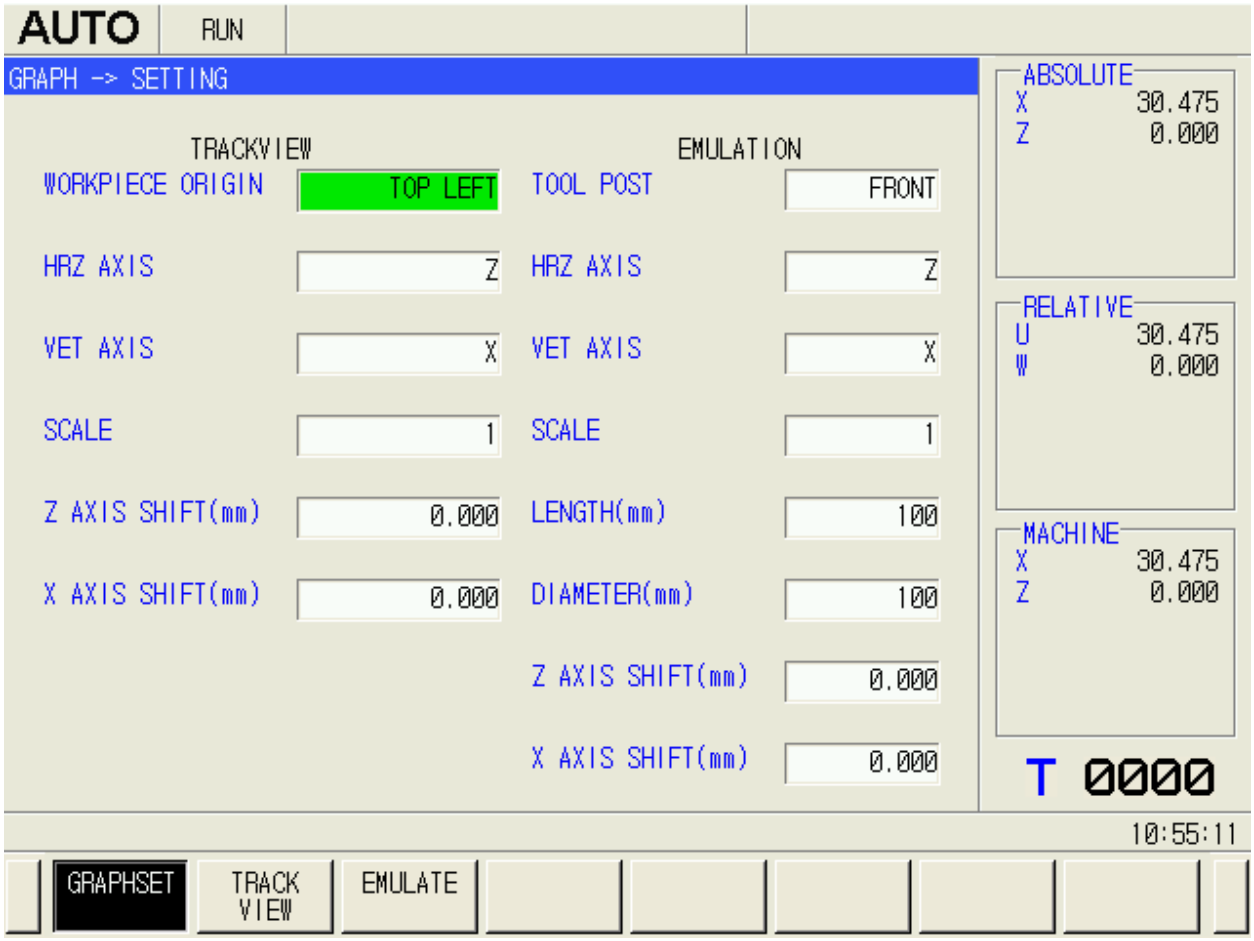


Fig.3-47

On the window, the path and the simulation parameters can be set.

Firstly, set the horizontal and vertical axes of the graph, and set the offset of the coordinate axis and the magnification of the graph; if the simulation graph is required, set the simulation horizontal and vertical axes, the length and the diameter of the processing work piece and the magnification of the simulation graph.

In the right column, it displays the current absolute position coordinate and the relative coordinate position value and the tool number used in the currently running program at the same time.

Press ⇧ or ⇩ to switch between items; in MDI mode, press the numerical and backspace keys to rewrite the graph parameter and input the rewritten value, and press INPUT to confirm the setting is completed. About the details, refer to chapter 8.1.

3.6.2 Processing graph path

On the graph window, press TRACK VIEW to access the path window and it is shown as the following graph 1:

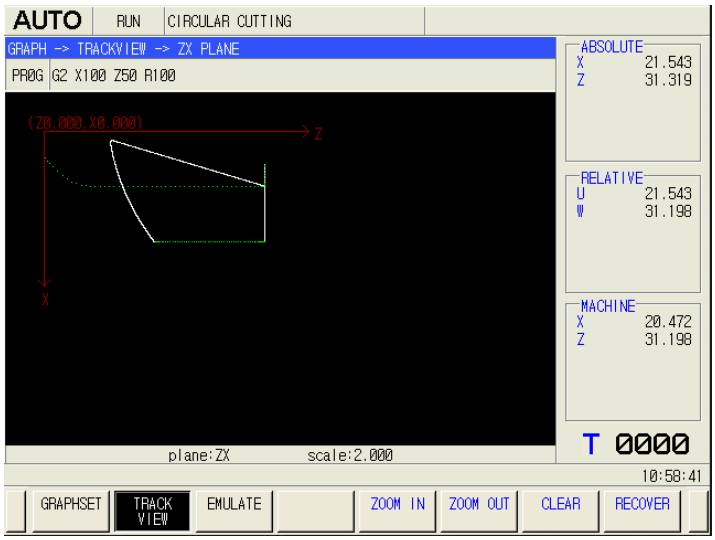


Fig. 3-48

In the figure, at the bottom of the path screen, it displays the coordinate level of the present path and the scaling of the path graph.

In the right column, meanwhile, it displays the current absolute position coordinate and the relative coordinate position value and the tool number used in the currently running program.

Then, the graph can be zoomed in and out and the path can be cleared, and press ⇧, ⇩, ⇦ or ⇨ to move the graph up, down, right or left.

Note: The name for each axis is set by parameter #1020, and the names are set in the different letters.

3.6.3 Simulation graph

On the graph window, press EMULATE to access the simulation graph window and it is shown as Fig.2:

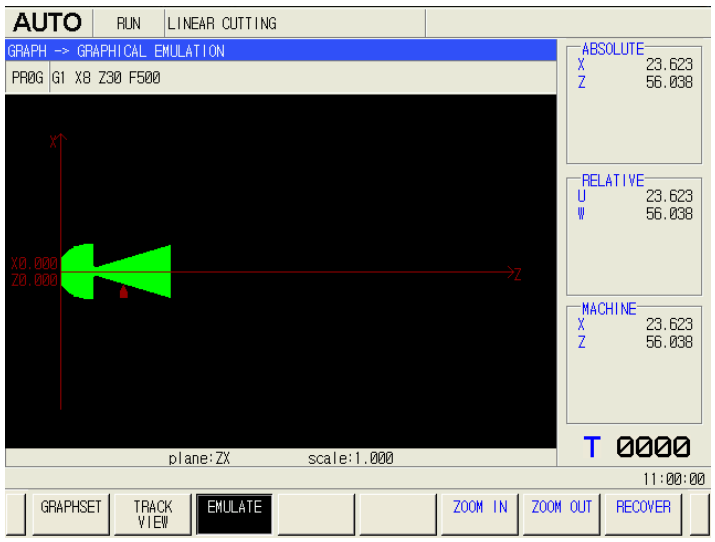


Fig. 3-49

In the figure, at the bottom of the simulation graph screen, it displays the coordinate level of the present simulation graph and the scaling of the simulation graph.

On the simulation graph window, only the graph simulation message of XZ level is displayed.

Then, the graph can be zoomed in and out and the path can be cleared, and press ⇧, ⇩, ⇦ or ⇨ to move the graph up, down, right or left.

Note: The name for each axis is set by parameter #1020, and the name can be set in different letters.

3.7 Help Windows

Press HELP to access help window shown in Fig. 3-50. It mainly includes the help of operation, programming, alarm and parameter windows, and check the content on the windows through pressing the corresponding soft keys:

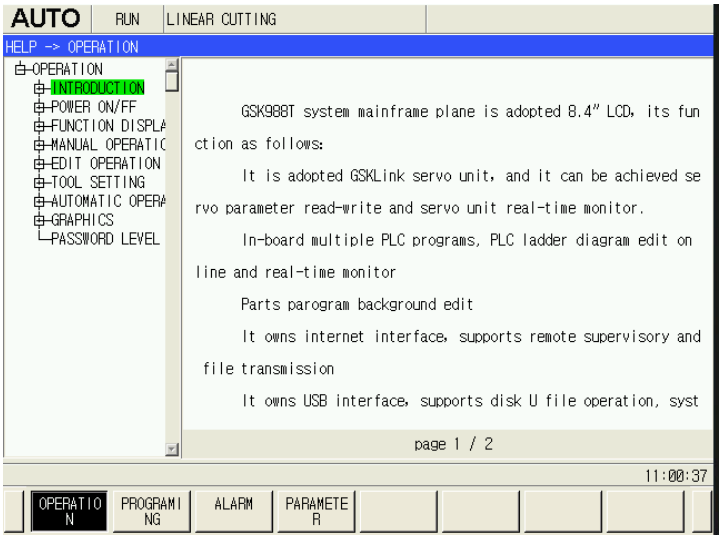


Fig.3-50

Each window is separated as two parts, the left column and the right relative content. The shortcut keys can be operated as below:

Content: Window up: turn to the last window in the content;
Window down: turn to the next window in the content;
Directory: Upward direction key: Check the last directory;
Downward direction key: Check the next directory;
Right direction key: Return to the previous directory;
Left direction key: Open the next directory;
Alter + window up key: turn to the last window in the directory;
Alter + window down key: turn to the next window in the directory;

Besides, there are the search functions in PROGRAMMING HELP, ALARM HELP, PARAMETER HELP window, the rapid search can be executed correspondingly by inputting the code, the miscellaneous function, the alarm number, the parameter number. Shown in Fig. 3-51, press SERACH in PROGRAMMING HELP window , input G01 in the dialog box and then press ENTER to directly find G01 code help shown in Fig.3-52:

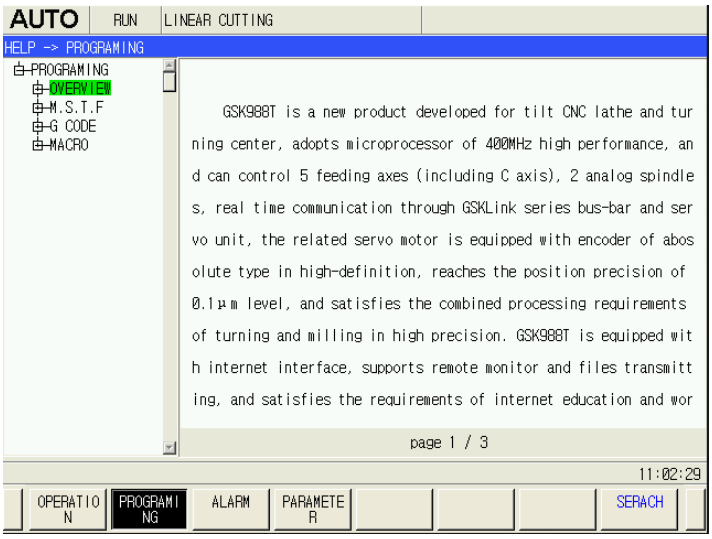

AUTO
RUN
LINEAR CUTTING
HELP -> PROGRAMING
PROGRAMING
OVERVIEW
M.S.T.F
G CODE
MACRO
GSK988T is a new product developed for tilt CNC lathe and turning center, adopts microprocessor of 400MHz high performance, and can control 5 feeding axes (including C axis), 2 analog spindles, real time communication through GSKLink series bus-bar and servo unit, the related servo motor is equipped with encoder of absolute type in high-definition, reaches the position precision of 0.1μm level, and satisfies the combined processing requirements of turning and milling in high precision. GSK988T is equipped with internet interface, supports remote monitor and files transmitting, and satisfies the requirements of internet education and wor
page 1 / 3
11:02:29
OPERATION
PROGRAMING
ALARM
PARAMETER
SERACH

Fig.3-51

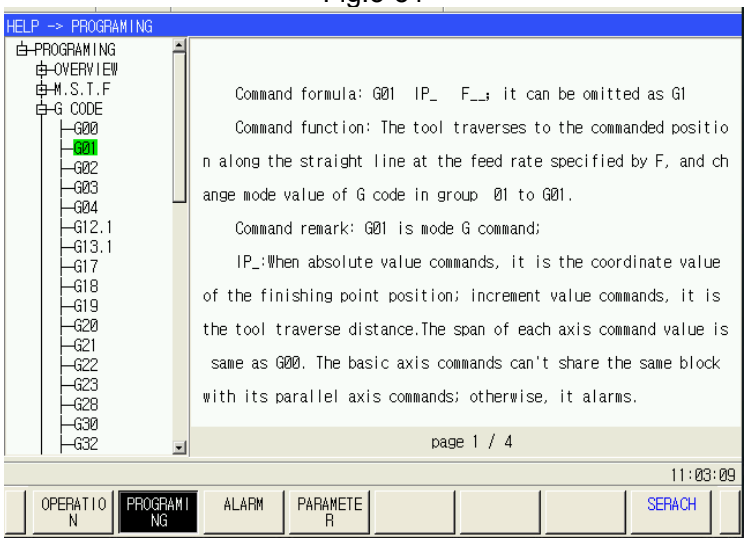

HELP -> PROGRAMING
PROGRAMING
OVERVIEW
M.S.T.F
G CODE
G00
G01
G02
G03
G04
G12.1
G13.1
G17
G18
G19
G20
G21
G22
G23
G28
G30
G32
Command formula: G01 IP_ F__; it can be omitted as G1
Command function: The tool traverses to the commanded position along the straight line at the feed rate specified by F, and change mode value of G code in group 01 to G01.
Command remark: G01 is mode G command;
IP_:When absolute value commands, it is the coordinate value of the finishing point position; increment value commands, it is the tool traverse distance.The span of each axis command value is same as G00. The basic axis commands can't share the same block with its parallel axis commands; otherwise, it alarms.
page 1 / 4
11:03:09
OPERATION
PROGRAMING
ALARM
PARAMETER
SERACH

Fig.3-52

Chapter Ⅳ Editing and Managing a Program

On the program window, the program can be created, selected, rewritten, copied and deleted, also imported and exported.

To prevent the programs are rewritten and deleted by accident, the program switches are set in GSK988T. Before rewriting the program, the program switches must be on. About the setting of program switches, refer to chapter 3.4.2.1.

Note: Create a file 'NCPROG' in the U disk, take the program into the file. At the moment, the operations in the U disk cataloge are consistent with those of the local catalog in the program page. Refer to the operations of local catalog when using the U disk catalog.

4.1 Searching, Creating, Executing and Opening a Program

4.1.1 Searching a program

（1）Press PROGRAM and LOCAL to enter PROGRAM window shown in Fig. 4-1:

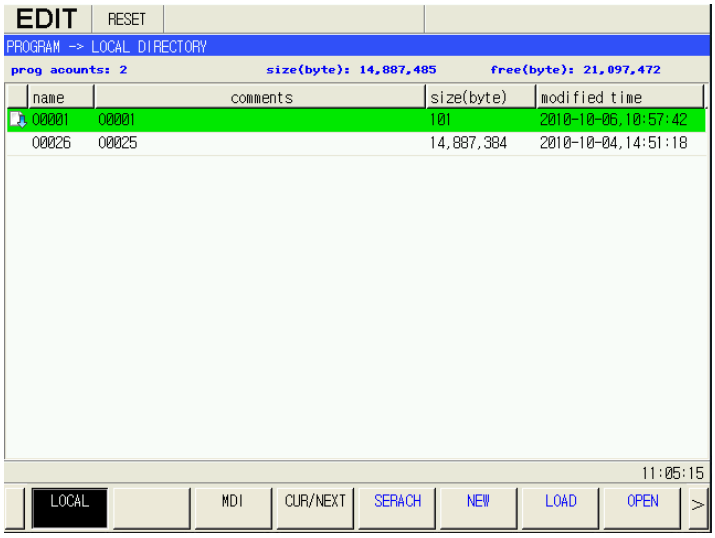


Fig. 4-1

（2）Press SERACH to enter the search window in PROGRAM window.

（3）Input the program name which is searched in

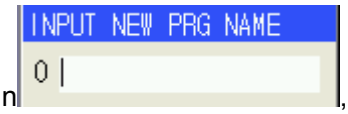
. For example, input 0005, press OK and the cursor directly skips to the program O0005. For example, the input does not exist in the CNC, the system at in the lower left corner prompts The file does not exist .

4.1.2 Creating a program

Only when the operation authority is above level (4), can the program be created and edited.

（1）Firstly press PROGRAM and then press LOCAL to access the program windows, which is shown in Fig.4-1:

（2）On the program window, press NEW to access the creating window, which is shown in Fig. 4-2:

（3）Input the new program name in 

, for example, input 0123, press NEW to access O0123 program editing window, which is shown in Fig.4-2:

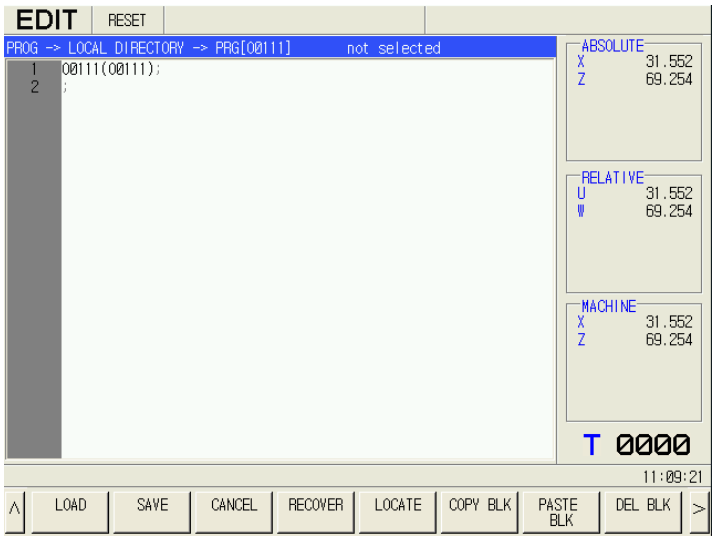


Fig. 4-2

4.1.3 Executing a program

（1）In Edit mode, press PROGRAM to enter PROGRAM window.

（2）In program window, press , to move the cursor to select the program name. Or press SERACH to search the program name which requires to run. The selected line is displayed against a green backdrop shown in Fig.4-3:

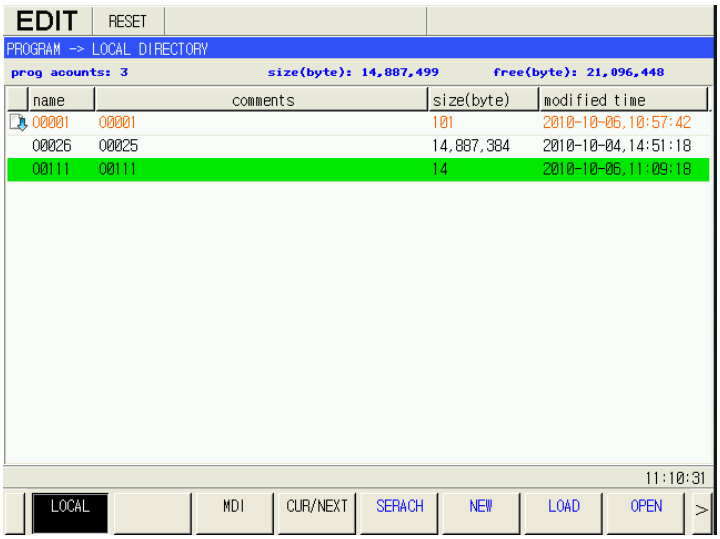


Fig.4-3

（3）In reset mode, press LOAD, and the selected program is loaded to the block area of the position window, which becomes the current executable program, the display window skips to the position window, at the moment, the system switches to Auto mode, press START, and a program can run.

4.1.4 Opening a program

（1）Press PROGRAM and then press LOCAL to access the program windows, which is shown in Fig.4-3:

（2）In the program windows, press , , or to select the program to be opened; or press SERACH to search, and input the program name to be opened, and then press OK to search, and the cursor positions in the program name, the background of the selected program name changes into green-based color, such as O0001 shown in Fig.4-3:

（3）Press OPEN to open the codes of the selected program in the screen, which is shown in Fig.4-4:

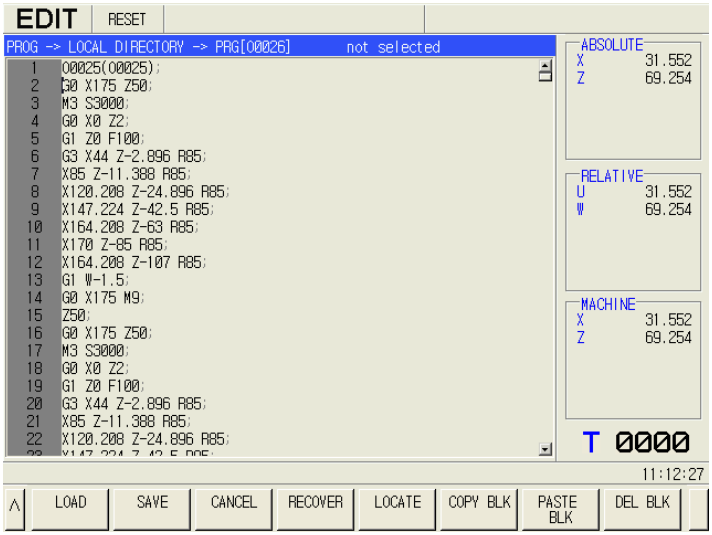


Fig.4-4

Then, the current program can be edited and rewritten, but when the program is being executed, it must be edited in the editing mode.

Note: When No.3404.6 is 0, the program must has the end code M02, M30, M99, otherwise, the system prompts the mistakes when PRG CHECK **is pressed to check the program, and the alarm occurs when a program runs.**

4.2 Renaming, Outputting, Deleting and Arraying Programs, Saving a Program as

In program window, press > to switch the window including renaming, deleting, outputting programs and saving a program as, which is shown in Fig.4-5:

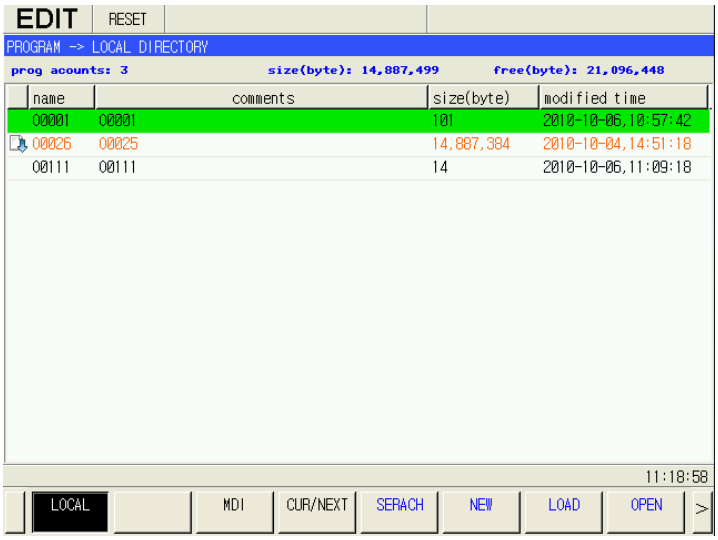


Fig.4-5

4.2.1 Renaming a program

In PROGRAM window, press ⇧, ⇩ to move the cursor to select a program, press

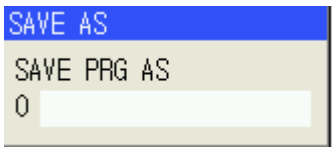
SAVE AS to rename the selected the program. Input a new program name in the dialog box

, press OK, and the selected program is renamed as the input new program name and the system returns. Press CANCEL to cancel the rename operation and the system returns to the previous menu.

Note 1: The file which has been loaded or is running cannot be renamed.
Note 2: Only when the operation authority is equal to or more than level [3], can renaming a program be executed.

4.2.2 Saving a program as

In PROGRAM window, press ⇧, ⇩ to move the cursor to select a program, press RENAME to save the selected program as another name. Input a new program name in the dialog box



, press OK to save the program as. For example, input 2222, press OK, and No.00011 program is saved as O2222, and the cursor skips to the new program name, which is shown in Fig. 4-6:

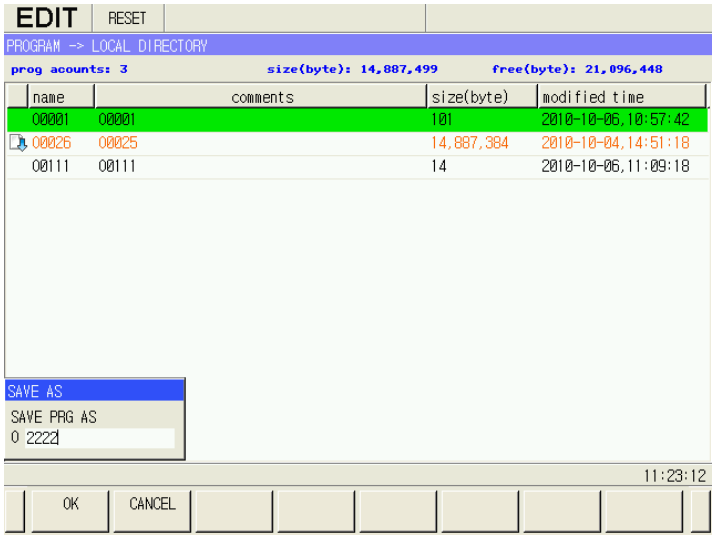


Fig.4-6

Note: Only when the operation authority is equal to or more than level [4], can saving a program as be executed.

4.2.3 Deleting a program

（2）In PROGRAM window, press , , or to select the program to be deleted, the selected program is against the green backdrop.

（3）Press DELETE to delete the selected program.

4.2.4 Outputting a program

When the system USB interface has the U disk, the following is shown:

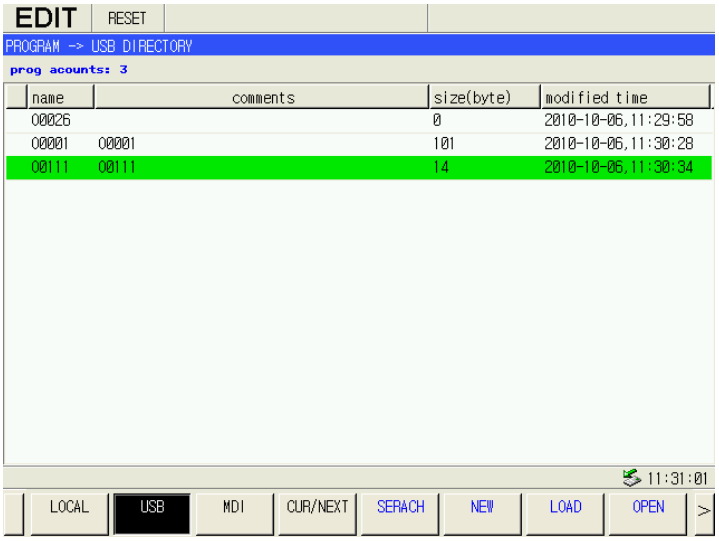


Fig.4-7

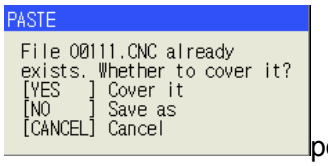
Press OUTPUT to copy the program in an U disk directory to the local directory, vice versa. The detailed steps of the program in the U disk being copied to the system are shown below:

(1) Press USB to access the directory of the U disk;

(2) Press or to select the program to be copied; press OUTPUT to copy the selected program to the local directory;

(3) Press LOCAL to access the system program directory;

(4) Press or to select the program to be copied; press OUTPUT to copy the selected program to the U disk;

(5) When the copied program exists, a dialog box 

pops-up: PROGRAM EXISTS, RECOVER?

(6) Press “yes” to replace the existed program; press “no” and the program can be saved as another name, press “cancel” to cancel the operation.

Note 1: Only when the operation authority is above level [3], can the copying and pasting be executed.

Note 2: When the output file is too big and the copy time is too logn, the system displays the processing, and the user can switch the page and does operations in the other pages.

4.2.5 Arraying programs

In program window, press [>] and the user can view [SORT BY NAME], [SORT BY SIZE], [SORT BY TIME] which can make the programs to be displayed orderly.

Press [SORT BY NAME], [SORT BY SIZE], [SORT BY TIME] repetitively, and the sort order of programs in all types can be switched between the positive-sequence and the inverted order.

4.3 Editing and Rewriting a Program

4.3.1 Editing a program

Creating a program based on Chapter 4.1.2, which is shown in Fig.4-8:

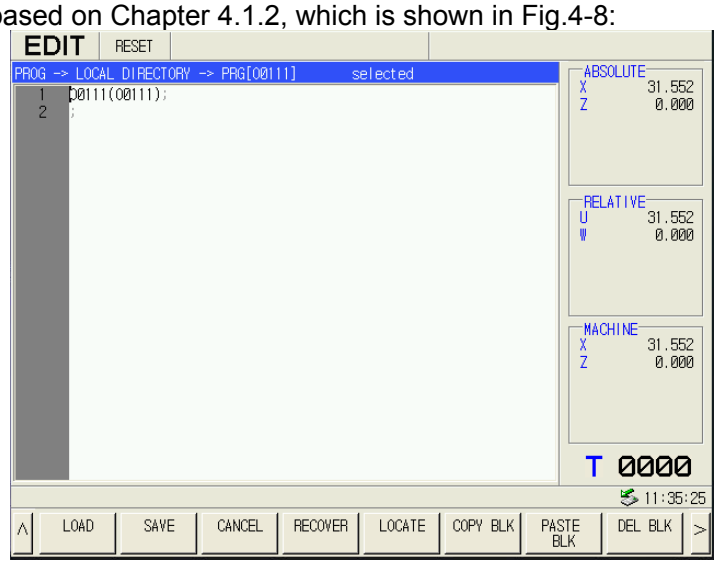


Fig.4-8

Edit a program based on the commands of GSK988T Programming User Manual.

- **Soft key introduction in edit window**

[LOAD]:After the current program is edited, pressing it can make the program in the executable state, at the moment, the window skips to the position window, and the selected program to be loaded is displayed in the block column of the position window. press [AUTO] and the system enters Auto mode; press [CYCLE START], and the system executes the loaded program.

[SAVE]:save the program being saved currently.

CANCEL:pressing the key can cancel the previous step of the program being edited. (cancel up to the last edited 10 steps) .

RECOVER:pressing the key can recover the previously cancelled program.

LOCATE:pressing the key can rapidly position to the specified line exactly.

COPY BLK:copy the block where the current cursor is.

PASTE BLK:in the place where pressing it can paste the previously copied block.

DEL BLK:delete the block where the current cursor is.

Press > , and SEARCH and PRG CHECK appear in the current window.

SEARCH:using it can rapidly find the character string, and positions the cursor to the behind of the searched character string. Select the three search mode FROM TOP, NEXT and PREV in the course of search.

PRG CHECK:After a program is edited, pressing PRG CHECK can check whether the program has mistakes, if have, there is a prompt below screen, please refer to the prompt, check and rewrite the program.

Note 1: When No.3404.6 is 0, the program ends with M02, M30, M99, otherwise, the system prompts the mistake when PRG CHECK is pressed to check the program, and the alarm occurs when the program runs.

Note 2: A big file cannot be edit (it exceeds 10 thousand lines).

Note 3: Besides manually saving programs, the system can automatically save them in the course of edit every 30 seconds.

4.3.2 Rewriting a program

（1）Open a program based on Chapter 4.1.4;

（2）Press , , , to move the cursor to the required line to rewrite; press , to move the cursor the required character to rewrite; also using and can find the required block and character to rewrite;

（3）Press the address, digital key on the edit keyboard to input the program code to rewrite;

（4）Press to delete the previous before the character where the cursor is;

（5）Press to delete the one following the character where the cursor is;

（6）Press to save the currently rewritten program.

4.3.3 Shortcut key

The system has some shortcut keys to conveniently edit and rewrite programs in the course of editing programs.

- ✓ **Debugging the cursor**

  Simultaneously press and to move the cursor the file header;

  Simultaneously press and to move the cursor the end-of-file;

  Simultaneously press and to move the cursor the line home;

  Simultaneously press and to move the cursor the line end.

- ✓ **Selecting a program**

  Press + , , , to move the cursor to the one behind the command to copy, at the moment, a block displayed in invert color is selected again.

- ✓ **Deleting a block**

  After a block is selected, is pressed, i.e., the deletion operation is completed;

- ✓ Copying a block;

  Simultaneously press and to copy the selected block;

- ✓ Cutting a block;

- ✓ Simultaneously press and to cut the selected block;

✓ Pasting a block;

Simultaneously press CHANGE and Q N to past the copied or cut block.

4.4 Block Comment

When a block is commented, “EOB” is pressed with “;” behind the block, the content following “;” is the comments.

Example:

O0001；

G50 X0 Z0； set the coordinate zero;

G00 X100 Z100； rapid traverse to the position X100, Z100；

M30；

In the above program, the comment is added to the 2nd and the 3rd block, among which the content following the 1st semicolon is the comment, and the 2nd is the block’s end character which is automatically added after a block is completed to press INPUT the key.

Note: The Chinese comment must be edit by a PC because the system does not support Chinese input.

4.5 Generating a Block Number

In the program, the block number can be edited or not edited; the program is executed based on the editing sequence of the block (except calling).

In the setting windows, CNC setting window, when “auto generating number” switch is off, CNC can’t auto generate the block number, the block number can be edited manually during programming.

In the setting window, CNC setting window, when “auto generating number” switch is on, CNC auto generates the block number; during editing, press INPUT to enter a new line and auto generate the number of the next block, the increment value of the block number is set by CNC Value parameter #3216.

4.6 Background Editing a Program

In Auto or DNC mode, press PROGRAM to enter the program window, at the moment, the user can open the program to edit or create a program to edit, and the operations is the same those of the above.

Note 1: The user cannot edit the program which is running currently;

Note 2: In Auto or DNC mode, the user cannot press RESET when the background editing program is executed, otherwise, the running program resets to stop.

Chapter Ⅴ Manual Operation

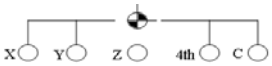
5.1 Manual Reference Position Return

There is a specified point in CNC machine, which can set the position of the machine working table. The specified point is taken as the reference position, in the position, the tool is changed and the coordinate system is set. After connecting the power supply, the tool traverses to the reference position. Manual reference position return is to use the switch and the button on the panel to traverse the tool to the reference position.

GSK988T system has three kind of reference position return mode: zero return with a dog, zero return without a dog and absolute encoder zero return.

- Setting a reference position with a dog:

When DLZx(No.1006 Bit 1) is set to 0, the reference position setting with a dog is invalid(i.e. the reference position setting with a dog is valid), a deceleration switch must be installed on the machine to realize the reference position return.

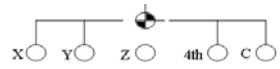
Process: The tool traverses in the direction specified by ZMI(No.1006 Bit5), to the deceleration point at the rapid traverse speed, and then at the FL speed to the reference position. The reference position return completion light(LED) 

lights and the reference position return ends and the system automatically sets the coordinate system.

Note: The rapid traverse speed, the rapid traverse override F0, the reference position return FL speed for each axis are separately set by the parameters No.1420, No.1421, No.1425.

- Setting reference position without a dog:

When DLZx(No.1006 Bit 1) is set to 1, the reference position setting without a dog is valid. The reference position return can be completed without a deceleration switch installed on the machine.

Process: The tool traverses in the direction specified by ZMI(No.1006 Bit5) when the machine is turned on every time and the reference position return is executed, and after the system has checked the 1st PC signal of the motor, the reference position return completion light(LED) 

lights and the reference position return ends and the system automatically sets the coordinate system.

Note: Because there is no dog, the system checks the first PC signal of the encoder as the position of the reference position, the set reference positions every time are different, and so the tool offset must be set again after the reference position is set in the mode.

- **Reference position setting with an absolute encoder**

When the machine is allocated with the absolute encoder and the reference position return function with an absolute position encoder is valid, and the system has not created the reference position, the reference position return with the absolute position encoder must be executed. After the tool returns to the reference position, the reference position return completion light LED lights and the system automatically sets the coordinate system.

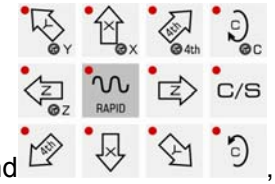
The reference position return steps:

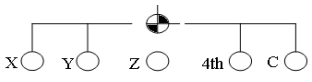
1. Press

and it is one of the mode selecting switches;

2. To decelerate, press one of the rapid traverse override switches

;

3. Press the feeding axis corresponding to the reference position return and 

, execute the reference position return. The tool traverses to the deceleration point at the rapid traverse speed and then traverse to the reference position at FL speed set by parameter. After the tool returns to the reference position, the reference position return finish indicator (LED) 

is on;

4. Execute the same operations for the other axes.

Note 1: Manual reference position return can only return to the 1st reference position; after the manual reference position return finishes, the coordinate system is auto set.

Note 2: Once the reference position return finishes, "reference position return finish" indicator is on, the machine doesn't move anymore until the reference position return switch is cut off.

Note 3: When "reference position off" or "during emergency stop", the reference position return finish indicator is off.

Note 4: The direction for each axis reference position return is set by the 5th bit of parameter #1006.

Note 5: Setting the 2nd bit of parameter 1404: After set the reference position, manually return to reference position, and moves to the reference position at the rapid feedrate or manual rapid feedrate.

Note 6: After the system reference position of the absolute encoder is set, auto set the coordinate system after power on again, and it doesn't require reference position return. But the non-absolute encoder system requires executing the reference position return after power on again.

The above is just one example; refer to the manual provided by the machine manufacture during the actual operation.

5.2 Manual Feed

In Manual mode, press the feeding axes and direction selection switches on the machine panel, the machine moves along the selected axis.

Each axis manually continuous feedrate is set by parameter (#1423), and each axis manual continuous feedrate can be adjusted through manual continuous feedrate override dial.

Feedrate dial

Press

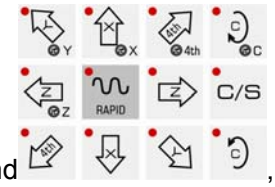
and the machine moves at the rapid traverse speed set by #1424 parameter, no matter where is JOG feedrate override dial, and the function is called the manual rapid traverse.

During the manual operation, many axes can move at the same time.

JOG feed steps:

1. Press

and it is one of mode selection switches,

2. Press feeding axis and 

, the machine moves along the corresponding axis in the corresponding direction. When the switch is pressed, the machine moves at the feedrate set by parameter (#1423); once the switch is released, the machine stops feeding;

3. Manual continuous feedrate can be adjusted through the manual continuous feedrate override dial;

4. If the feeding axis, the direction selection switch and

are pressed meanwhile, the machine moves at the rapid traverse speed, and

can be selected and valid during the rapid traverse;

The above is just one example; refer to the manual provided by the machine manufacturer during actual operation.

Note 1: Acceleration/deceleration;
Manual rapid traverse speed, the time constant and the mode of acceleration/deceleration can be set by parameter 1610 and 1624.

Note 2: Changing the mode:
During JOG feeding, when the mode is switched into the other mode, JOG feeding becomes invalid. To make JOG feeding valid, firstly access JOG feeding mode, and then press feeding axis and mode selection switch.

Note 3: Rapid traverse before the reference position return:
If the reference position doesn't return after connecting the power supply, even press "rapid traverse" button, it can't run; while remain manual continuous feeding traverse. The function can be set by parameter RPD (0 bit of #1401).

Note 4: In manual mode, whether JOG override is valid, which is set by the 2nd bit of parameter #1402; when it is invalid, the override is fixed as 100%.

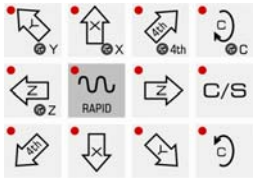
5.3 Increment Feeding

In JOG mode, during MPG or manual feeding, whether the increment feeding is valid, this is set by parameter JHD (0 bit of #7100). The corresponding relation is shown as below:

	JHD=0		JHD=1	
	JOG mode	MPG mode	JOG mode	MPG mode
JOG feeding	O	×	O	×
MPG feeding	×	O	O	O
Increment feeding	×	×	×	O

O: Valid

×: Invalid

In increment mode, press feeding axis and 

on the machine panel, the machine moves one step in the selected axial direction. The minimum distance which machine moves is the minimum input increment, and each step can be 1 time, 10 times or 100 times of the minimum input increment.

Increment feeding steps:

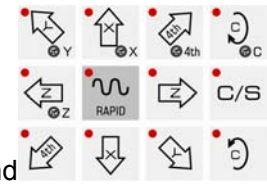
1. Press

to select MPG mode;

2. Press

to select the amount of movement of each step; moreover, the distance of each step selected by

is rewritten by parameter 7113 and 7114;

3. Press the feeding axis and 

, the machine moves along the selected axial direction. Press the switch for one time, it moves for one step. Its feedrate is same as the manual continuous feedrate.

4. Press rapid traverse switch when the feeding axis and the direction selection switches are on, the machine moves at rapid traverse speed.

The rapid traverse override is valid during the rapid traverse.

Note: The minimum input unit (input) and the minimum command increment (output)are set by the 1st bit of parameter #1004. The minimum input increment is the minimum unit of the programmed amount of movement, the minimum command increment is the minimum unit of the tool traverse on the machine, and the two increments are represented by millimeter or inch.

5.4 MPG Feeding

Press

to access MPG mode, the appearance of MPG is shown as below:

MPG outside drawing

In MPG mode, the machine moves continuously through rotating MPG on the operational panel.

And press to select the movement axis. When MPG rotates one graduation, the minimum distance of the tool traverse is the minimum input increment. When MPG rotates one graduation, the tool traverse distance can be magnified 10 times or one of two overrides is set by parameters #7113 and #7114.

MPG feeding steps:

1. Press to access MPG mode;
2. Press to select the axis which is moved by one machine;
3. Press to select the override of the machine movement. When MPG rotates for one graduation, the minimum distance traversed by the machine is the product of the minimum input increment multiplying the current override. The override set by can be rewritten by parameters #7113 and #7114;

4. Rotating MPG machine moves along the selection axis, MPG rotates for 360° and the amount of the machine movement is that of 100 graduations.

MPG feeding direction is set by MPG rotation direction. Normally, MPG CW feeds positively, CCW negatively.

The above is just one example; refer to the manual provided by the machine manufacturer during actual operation.

Note 1: In JOG mode (JHD), MPG is valid;
In JOG mode, whether MPG can be used, which is set by parameter JHD (the 0 bit of #7100), when parameter JHD (the 0 bit of #7100) is set as 1, MPG feeding and increment feeding are both valid.
The corresponding relation is shown as the following list:

	JHD=0		JHD=1	
	JOG mode	MPG mode	JOG mode	MPG mode
JOG feeding	O	×	O	×
MPG feeding	×	O	O	O
Increment feeding	×	×	×	O

O: Valid
×: Invalid

Note 2: The commands of MPG exceed the rapid traverse speed (HPT);
The parameter HPT (the 4th bit of #7100) is stipulated as below:
Setting to 0: When the feedrate is limited by the rapid traverse speed, the impulse value exceeding the rapid traverse speed is invalid. (The amount of machine movement doesn't comply with MPG graduation).
Setting to 1: The feedrate is limited by the rapid traverse speed, and the impulse value exceeding the rapid traverse speed is valid, but it is accumulated in CNC. (Although MPG isn't rotated, the machine can't stop. After MPG stops, the machine still moves due to the effect of CNC pulse.) The allowable value of the memory capacity is set by parameter #7117, then, the part exceeding the memory capacity is ignored.
Note 3: Axial movement direction and MPG rotation direction:
Parameter HNGx (the 0 bit of #7102) switches into MPG direction which the tool traverses along the axis and it corresponds to MPG rotation direction.

Note 4: Quantity of MPG

The maximum 2 manual pulse generators can be connected, which is set by parameter #7110. The two generators can operate one selected axis meanwhile.

Chapter Ⅵ Auto Operation

6.1 Auto Running

The program should be saved in the memorizer in advance, when one program is selected and

is pressed on the machine operation panel, the program automatically runs and the cycle start indicator is on. During cycle, press

, auto running pauses. When

is pressed once more, auto running starts again. When

is pressed on MDI panel, auto running ends and resets.

6.1.1 Selecting the running program

（1）In auto or edit mode, press

to access the program windows.

（2）In the program windows, press
,
,
or
, and the cursor moves to select the program name, or press

to search the program name to run. The selected program line displays against a green backdrop, which is shown in Fig.6-1:

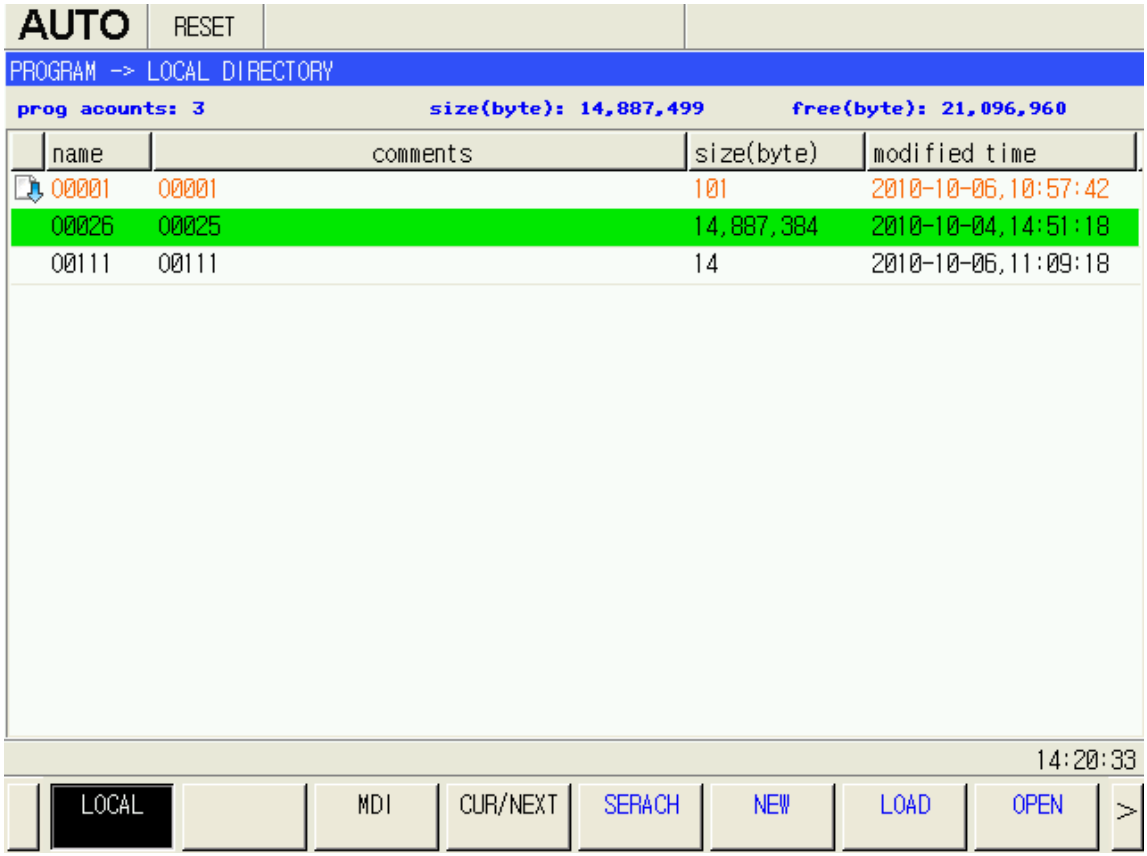


Fig.6-1

（3）During resetting, press LOAD, the selected program is uploaded into the block area in the position window and it can be executed, the current window switches into the position window, which is shown in Fig.6-2:

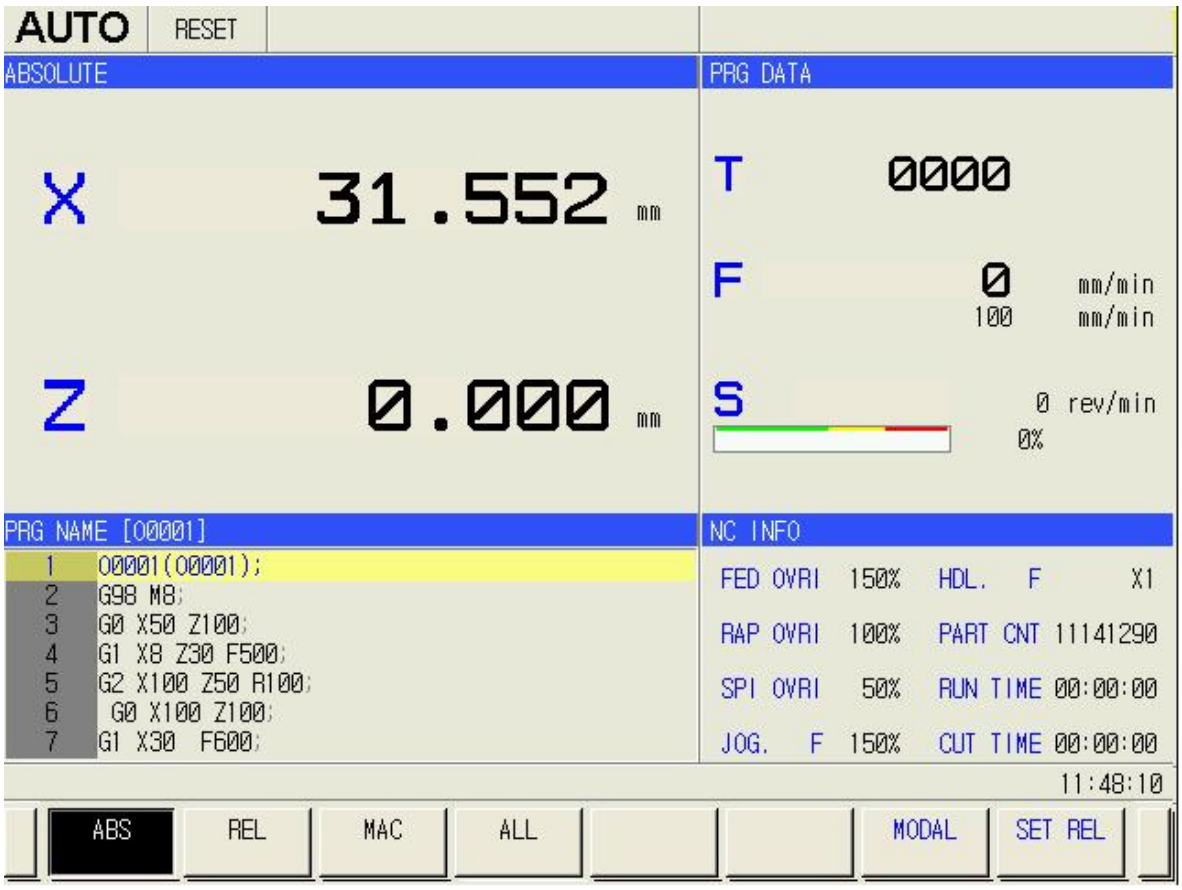


Fig. 6-2

Note: Only during resetting, the files can be uploaded.

6.1.2 Program running

1. Press AUTO to select the auto mode;

2. Press CYCLE START to start the program, and the program auto runs and the cycle starting indicator is on; while running ends, the indicator is off. When the last block specifies M99, it can return to the beginning of the program to run the program in cycle after running ends.

3. To stop during running or cancel the memorizer running, there are following methods:

1）Stop the memorizer running

Press FEED HOLD on the machine operation panel and its indicator is on, while the cycle start indicator is off. The machine responds as below:

a. When the machine is moving, feed running decelerates till stopping.

b. When pause (stopping) is being executed, the running stops.

c. When M, S and T functions are executed, the running stops after completing M, S and T functions.

When feed hold indicator is on, press

on the machine panel, the machine runs, again.

2）Stop the memorizer running

Press

on MDI panel, auto running ends and resets.

Note: The program runs from the line which the cursor is, before

is pressed, check whether the cursor is on the block to run.

6.1.3 Running from any block

1. In the above figure 2, press

to access the auto mode, press

to access the program window, press or to select the program content window: press or , the cursor moves toward the block to run; or on the program window shown as the above figure 1, press or to select the program to run, press

to access the program editing window, and then press or , the cursor moves toward the block to run, and then press

, it returns to the position window;

2. If the mode defaults (G, M, T and F commands) in the block which the cursor is, and the mode doesn't comply with that of the block, the next step can be operated only after the corresponding mode function is executed;

3. Press

to access the auto mode, and press

to start the program, the program begins executing from the selected block.

6.1.4 Skip

When a block is followed by "/", the

is pressed to start the skip mode, and the skip switch indicator lights,

is pressed and the block is skipped and is not executed. Taking example of the 4th line of the following program:

O0001；

G50 X0 Z0；set the coordinate zero;

G01 X100 Z100；rapidly traverse to the position X100, Z100；

/G0 X0 Z0；

M30；

The 4th line is skipped when

is press to run the program.

6.1.5 G31 skip

When G31 is edit before a block, and the external skip signal(X.3.5)is input during the course of G31 being executed, G31 running is interrupted to execute the next block. The function is used to the dynamic measure(such as milling machine) and the toolsetting measure of the workpiece dimension. Taking example of the 4th line of the following program:

O0002；
G31 Z200 F100；When the block is executed and the external skip signal (X3.5) is input, the block is interrupted and the next block is executed.
G01 X100 Z300；
……；
M30；

Note: The detailed use of G31 is referred to *PROGRAMMING*.

6.1.6 Stop auto running

The memorizer running can be stopped through the following methods: Command stopping or press the relative keys, which are on the machine operation panel, to stop.

- Command stopping (M00, M01, M02 and M30)

After executing the block with M00 or M01 (the selecting stop button on the panel is on), running automatically stops, the mode function and the state all are saved. Press

, the program continues to execute. When read in M02 or M30 (command at the end of the main program), the program running ends and resets.

The operations of different machines are not same; about the details, refer to the manual of the machine manufacturer.

- **Pressing relatives keys to stop**

1. During auto running, press

and the machine is shown as below:

（1）Machine feeding decelerates till stopping;

（2）The mode function and the state are saved;

（3）Press

, the program continues to execute.

2. Press

（1）All axes running decelerates till stopping;

（2）M and S functions output invalid (After pressing

, whether auto switch off signals of spindle CW/CCW, lubricating, cooling, etc is set by the parameter.)

（3）After auto running ends, the mode function is hold.

3. Press emergence stop button

During machine running, in the dangerous or the emergency case, press the emergency stop button (the external emergence stop signal is valid), CNC accesses the emergency stop, then the machine running stops immediately, all output is off, such as the spindle revolving and the cooling fluid, etc. Press the emergency stop button, the emergency stop alarm clears, and CNC resets.

4. Switching the operation mode

During auto running, switch into the reference position return, MPG/single or manual, the current block “pause” at once; during auto running, switch into edit or MDI mode, the running stops after running the current block.

Note 1: Confirm the trouble is shot before clearing the emergence stop alarm;
Note 2: Before power on or shutdown, press emergency stop button to reduce the electric shock to the equipment;
Note 3: After clearing the emergency stop alarm, return to the reference position, again to gurantee the correctness of the coordinate position.

6.2 MDI Running

6.2.1 Editing and running the program in MDI mode

（1）the program window, press

to access MDI mode, then the window is shown as below:

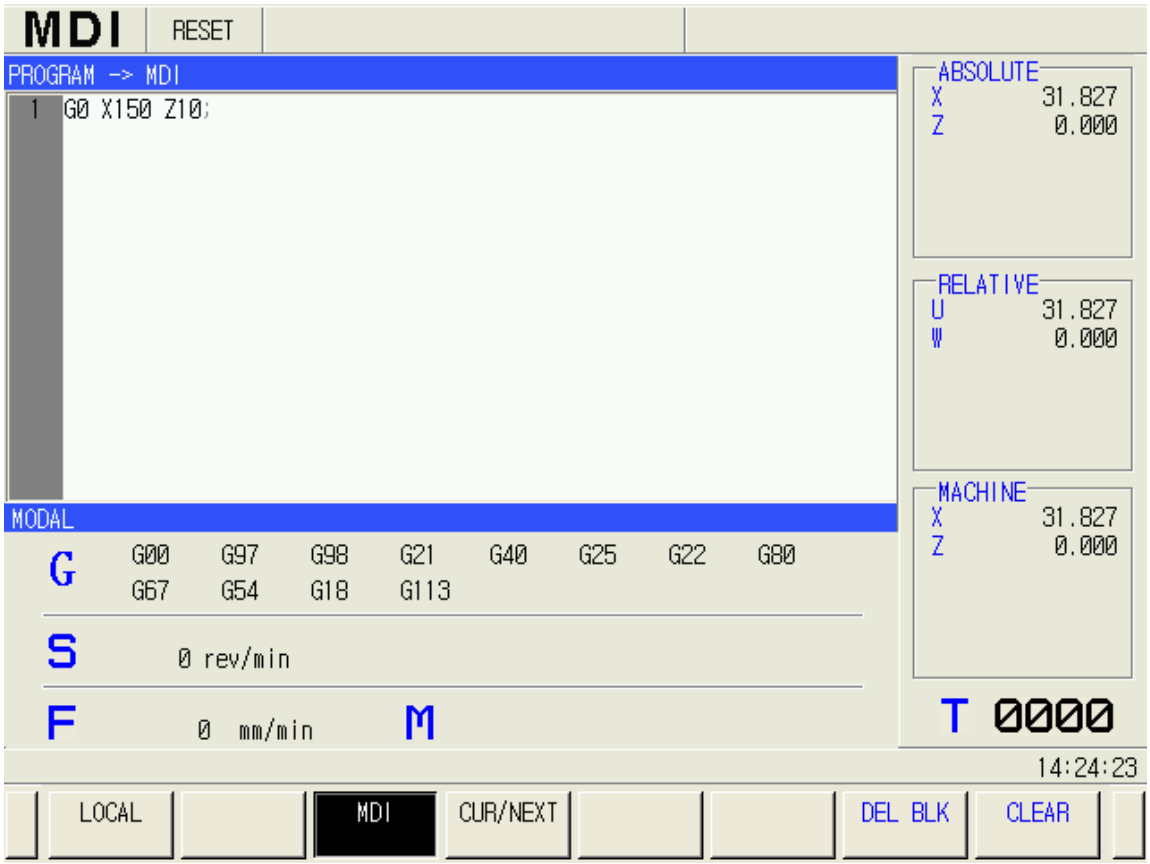


Fig.6-3

（2）The running block is input (maximum 10 lines) in the edit bar which is at the downside of the block (MDI); The editing method is similar with that of editing the common program. If the program is created in MDI mode, the characters can be rewritten and deleted. About editing the program, refer to chapter 5th.

（3）After the block is input, the cursor moves toward the beginning of the block, and executes. If the cursor is in somewhere of the program, the program is begun to execute. Press

, MDI command characters are executed from the line which the cursor is. When the program end codes (M02 or M03) are executed, the program running ends rather than return to the beginning of the program. After running the program, the system accesses the stop mode.

（4）During running, press

,

or the emergency stop button to stop MDI command characters.

Note 1: Deleting the program:

a. In MDI mode, press

to delete the block which the cursor is, press

to clear all the blocks in MDI edit bar.

b. When parameter MCL（NO.3203#7）is set as 1, press

and the program is auto cleared.

c. When parameter MER（NO.3203#6）is set as 1, in single block mode, after running the last block, the program is auto cleared.

Note 2: When MDI running stops, after editing,

is pressed to run again, the running starts from the position where the cursor is.

Note 3: The program which is created in MDI mode can't be saved.

Note 4: In MDI mode, the subprogram and the macro program can't be called.

6.2.2 Running from any block

In the position window, in MDI mode, press or , the cursor moves toward the block to run, press

to start the program and the program begins executing from the block which the cursor is.

6.2.3 Stop MDI running

MDI running can be stopped through the following methods: Command stopping or press the relative keys on the machine panel to stop.

The run stop in MDI mode is the same that in Auto mode, please refer to ***Chapter 6.1.4***.

6.3 DNC Running

988T is equipped with DNC function, and DNC communication software is connected with CNC, then the program is running in high speed and large capacity.

Press

on the machine panel, access DNC mode, after PC is ready, press cycle start key and start the program for DNC processing.

About the detailed method, refer to the introduction of DNC communication software.

1. Communication software GSKComm selects and opens the machine program.

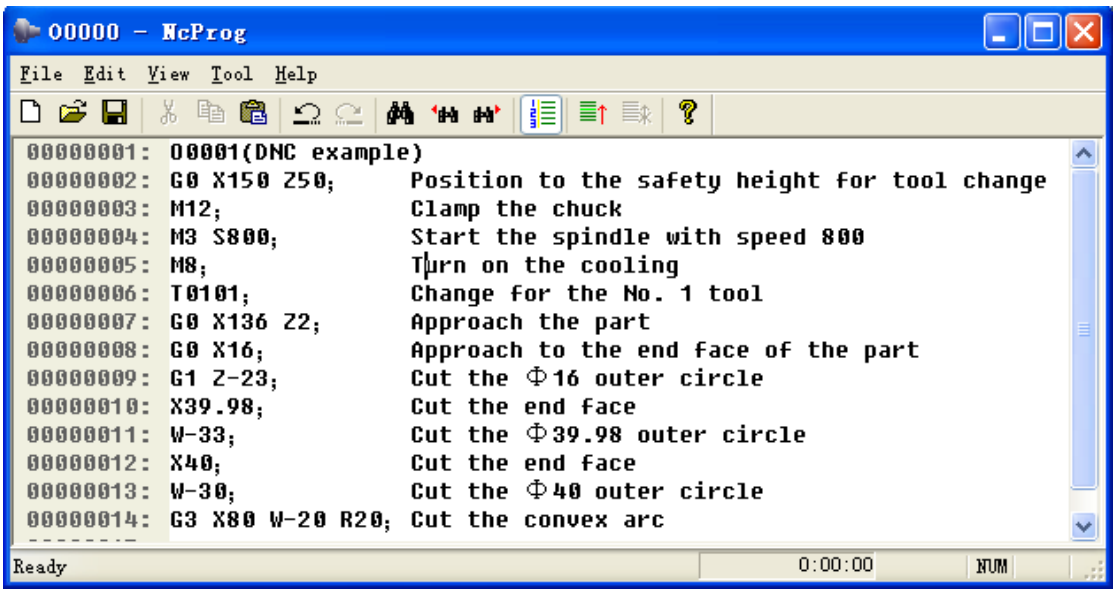

O0000 - NcProg
File Edit View Tool Help
00000001: O0001(DNC example)
00000002: G0 X150 Z50; Position to the safety height for tool change
00000003: M12; Clamp the chuck
00000004: M3 S800; Start the spindle with speed 800
00000005: M8; Turn on the cooling
00000006: T0101; Change for the No. 1 tool
00000007: G0 X136 Z2; Approach the part
00000008: G0 X16; Approach to the end face of the part
00000009: G1 Z-23; Cut the Φ16 outer circle
00000010: X39.98; Cut the end face
00000011: W-33; Cut the Φ39.98 outer circle
00000012: X40; Cut the end face
00000013: W-30; Cut the Φ40 outer circle
00000014: G3 X80 W-20 R20; Cut the convex arc
Ready
0:00:00
NUM

Fig.6-4

2. Connect CNC system.

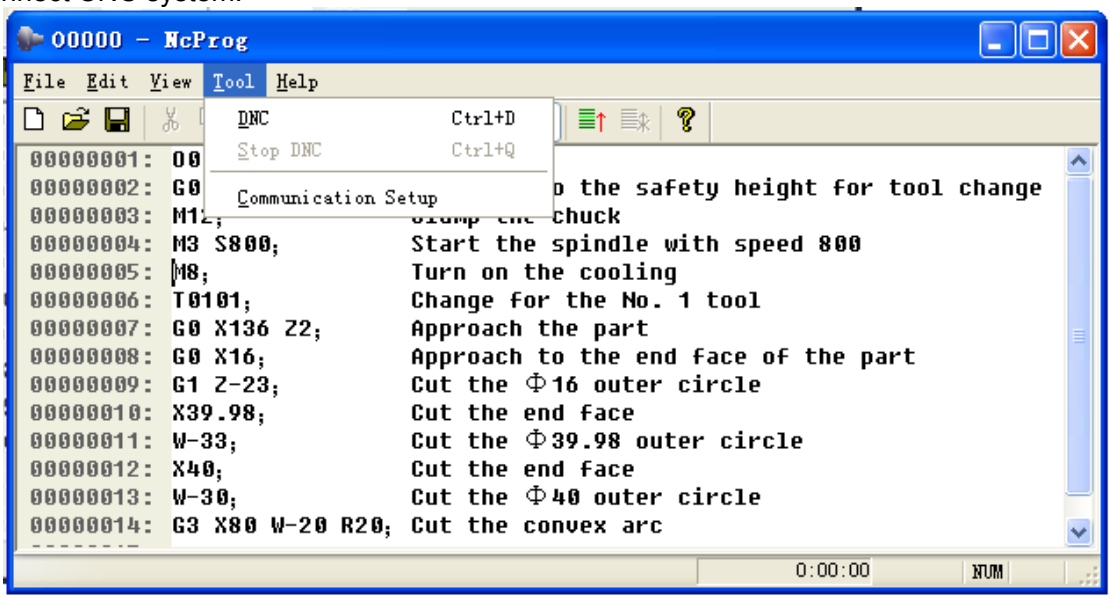

O0000 - NcProg
File Edit View Tool Help
DNC Ctrl+D
Stop DNC Ctrl+Q
Communication Setup
00000004: M3 S800; Start the spindle with speed 800
00000005: M8; Turn on the cooling
00000006: T0101; Change for the No. 1 tool
00000007: G0 X136 Z2; Approach the part
00000008: G0 X16; Approach to the end face of the part
00000009: G1 Z-23; Cut the Φ16 outer circle
00000010: X39.98; Cut the end face
00000011: W-33; Cut the Φ39.98 outer circle
00000012: X40; Cut the end face
00000013: W-30; Cut the Φ40 outer circle
00000014: G3 X80 W-20 R20; Cut the convex arc
0:00:00
NUM

Fig.6-5

3. Press

to select DNC mode:

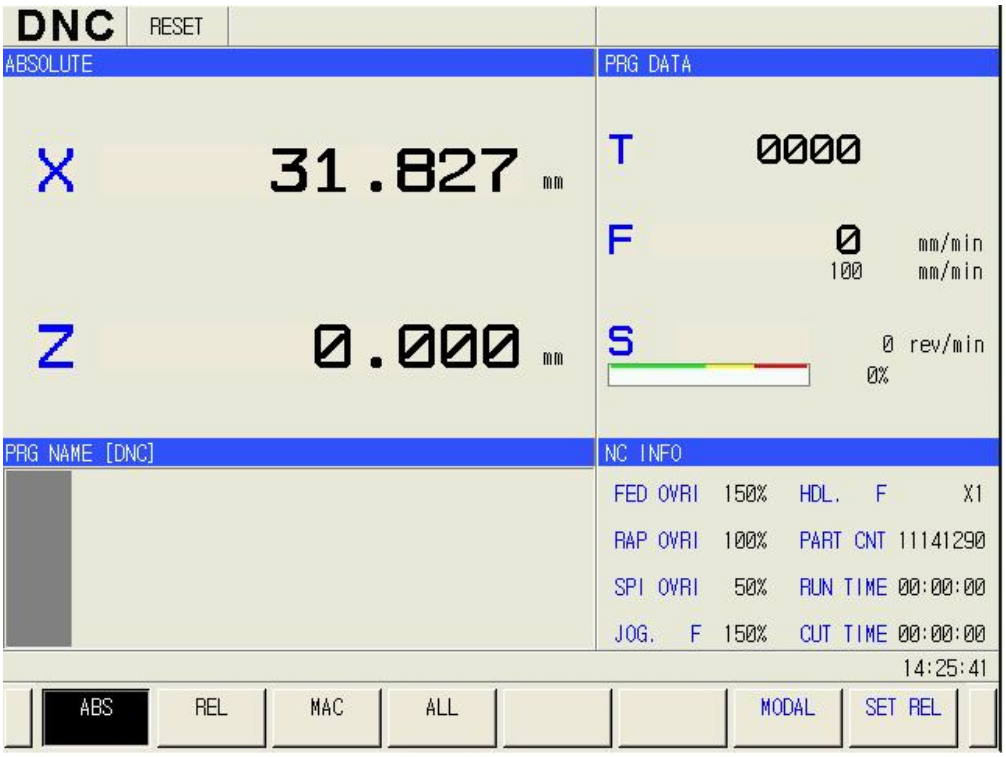


Fig.6-6

4. Press

, the program automatically starts and the cycle start indicator is on. After automatic running ends, the cycle start indicator is off.

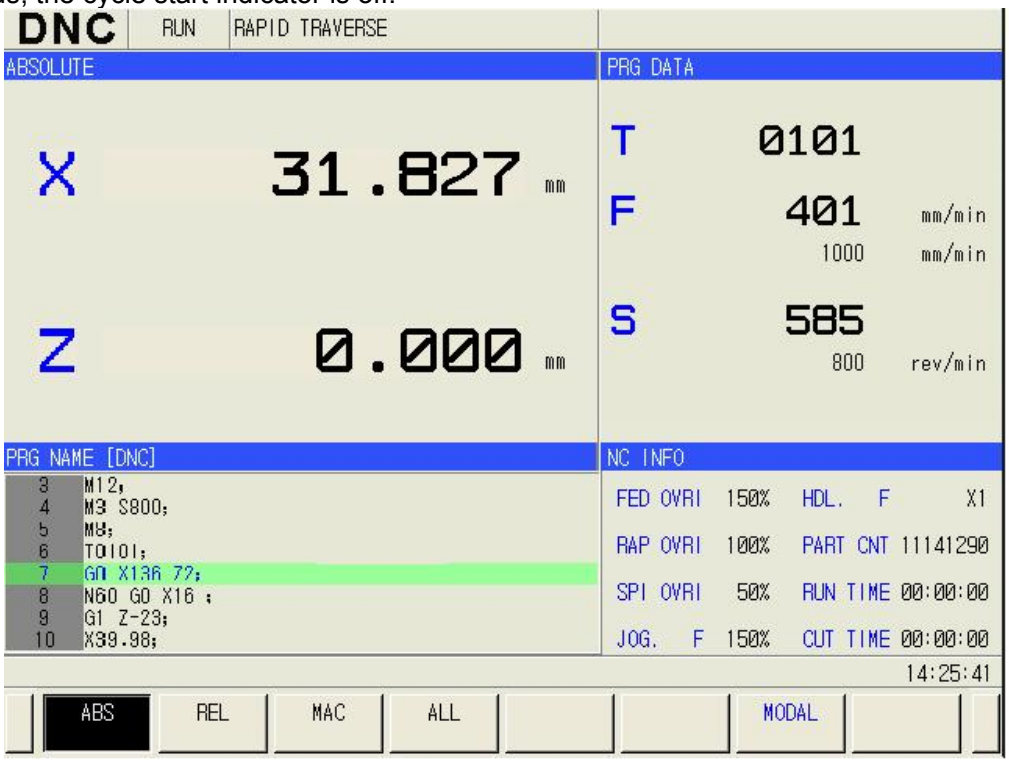


Fig.6-7

5. Stop during running

Press

, the feed hold indicator is on, while the cycle start indicator is off. The machine responds as below:

a. When the machine is running, the feeding decelerates till stopping.

b. When the pause (the tool stops running) is being executed, the running stops.

c. When functions of M, S and T are executed, running stops after completing the functions of M, S and T.

When feed hold indication is on, press

on the machine operation panel, the machine runs again.

6. Running end

Press

on MDI panel or DNC program executes M30 command, reset after running ends.

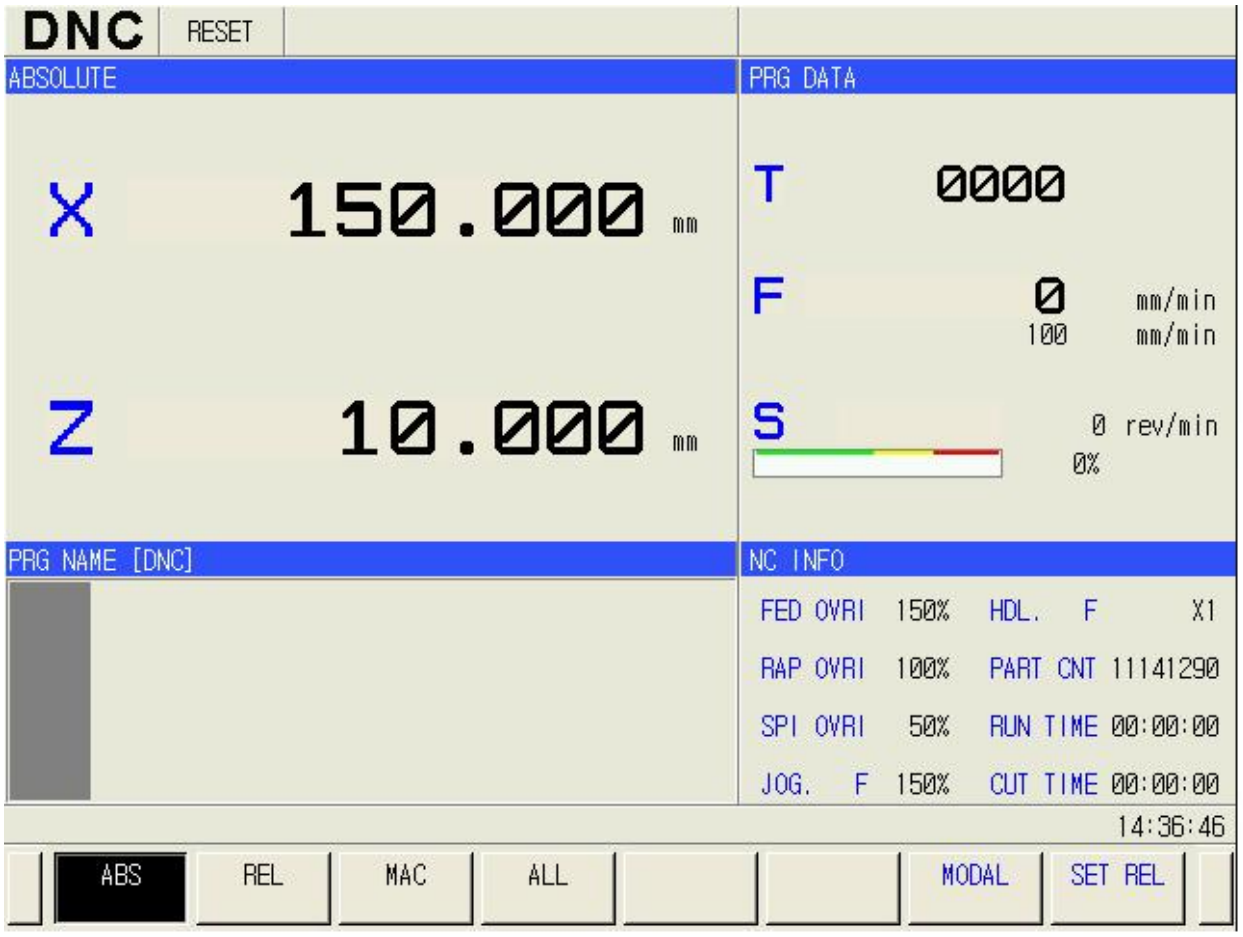


Fig.6-8

Note: In DNC program, the program calling and jumping commands can't be executed.

6.4 Auto Running Control

6.4.1 Machine and miscellaneous function lock

Use the machine lock and execute the machine program, but the machine remains still, only the tool position changing situation displays. All axes are locked, and the movement of all axes is stopped. Moreover, the locking miscellaneous function can lock the commands of M, S and T. Same as the machine lock, it's for checking the programs.

6.4.1.1 Machine lock

Execute the machine program, but the machine remains still, only the tool position changing situation displays, and then, the machine is locked to check the program. When the machine is locked, the movement of all axes is stopped.

Press

on the operation panel, the machine is still, but each axis position on the monitor is changing. About the machine lock, refer to the manual provided by the machine manufacturer.

Note1: Position relation between the work piece coordinate system and the mechanical coordinate system may be different before or after automatically use the machine lock. Then, the coordinate sets the commands or execute the manual reference position return to set the work piece coordinate system.

Note 2: When the machine is locked and G28 or G30 command is sent, the command can be received rather than move to the reference position and the reference position return indicator is off.

6.4.1.2 Miscellaneous lock

Locking the miscellaneous function can lock the commands of M, S and T. Same as the machine lock, it's for checking the program.

Press

on the machine operation panel, when M, S and T codes are invalid, they can't be executed. About the miscellaneous function lock, refer to the manual provided by the machine manufacturer.

Note 1: When the machine is locked, M, S and T commands can still be executed;
Note 2: Even the miscellaneous function is locked, commands of M00, M01, M02, M30, M98 and M99 (subprogram calling function) can be executed.

6.4.2 Dry run

Press

on the operation panel, the machine moves at the speed set by the parameter without considering the feedrate specified in the program, which can check the machine movement which the work piece unloads from the working table.

Steps of dry running:

During automatically running, press

on the machine operation panel, the machine moves at the feedrate set by the parameter, and the rapid traverse switch can change the feedrate. About the details of dry running refer to the manual provided by the machine manufacturer.

According to the rapid traverse switch and the parameter, the dry running speed change is shown as below:

Rapid traverse button	Program commands	
	Rapid traverse	Feeding
ON	Rapid traverse speed	Dry running speed *JVmax
OFF	Rapid traverse speed	Dry running speed *JV

JVmax: The maximum graduation value of the feedrate override

JV: The graduation value of the feedrate override

Note 1: The maximum cutting feedrate is set by parameter #1422;
Note 2: The rapid traverse speed is set by parameter #1420;
Note 3: The dry running speed is set by parameter #1410.
Note 4: The dry run speed is set by No. 1410.
Note 5: The dry run cannot be switched in Auto mode but can be switched in pause state.
Note 6: The dry run followed single block stop or pause is switched in G83/G85, but the actual cutting speed remains unchanged. After the cutting is completed, the speed can be switched. The dry run function cannot be activated or closed really and it is done after the single block stops or pauses. Even if the dry run is switched in the course of drilling, its speed remains the previous feedrate(it is dry run mode before cutting, it runs at the dry run speed, and reversely, does at the commanded speed), and the speed can be switched after cutting is completed.

6.4.3 Single block running

When execute the program at the first time, select the single block running to prevent the malfunction due to the programming mistakes.

In auto mode, the method of opening the single block switch is as below:

Press

to start the single block mode and the single block indicator is on. In single block mode, press

to execute one block, and then the machine stops; continue to execute the next block, press

again, repeatedly, until the program running ends. In the single block mode, check the program through executing the blocks one by one.

Steps of the single block running:

1. Press

on the machine operation panel, press

to execute one block in the program. After executing the current block, the machine stops;

2. Press the cycle start button and execute the next block; after executing the block, the machine stops.

Note 1: Reference position return and single block running: if commands of G28 and G29 are sent, the single block function in the intermediate point is valid.
Note 2: Subprogram block and single block running: with M98P_ or M99, or in G65 block, the single block stops.
Note 3: About executing the fixed cycle and multiply cycle in the single block mode, refer to the relative content in the command manual.

6.4.4 Feedrate override

The feedrate of programming can be decreased or increased through selecting the percent (%) on the override dial, which is for checking the program. For example, the machine move at 50mm/min when the specified feedrate is 100mm/min in the program and the override is set as 50%.

The steps of changing the feedrate override: before automatic running or during running, the feedrate override dial can be set as the expected percent (%).

Feedrate override button

The override can be specified from 0 to 150%. For some machine, the range is stipulated in the manual.

Override of the thread cutting: During the thread cutting, the override is invalid but the feedrate specified by the program is still valid.

6.4.5 Rapid traverse override

For the rapid traverse speed, there are four overrides (F0, 25%, 50% and 100%). The rapid traverse speed for each axis is set by parameter #1420; F0 is set by parameter #1421.

The step of changing the rapid traverse override: During the rapid traverse, select one override through pressing

.

The following types of the rapid traverse are valid and the rapid traverse override can apply to them:

1. G00 rapid traverse
2. Rapid traverse during the fixed cycle
3. Manual rapid traverse
4. Rapid traverse during the manual reference position return
5. Rapid traverse during G28 and G30

Chapter Ⅶ Tool Offset and Setting Tools

To simplify the programming, the actual position of the tool isn't taken into consideration during programming and GSK988T provides the methods of in-position tool-setting and trial cutting, etc, and get the tool offset Value through setting tools.

7.1 Setting the Tool Offset and the Wearing Values

7.1.1 Direct input method

（1）On the setting window, press

to access the tool offset management window, which is shown as below:

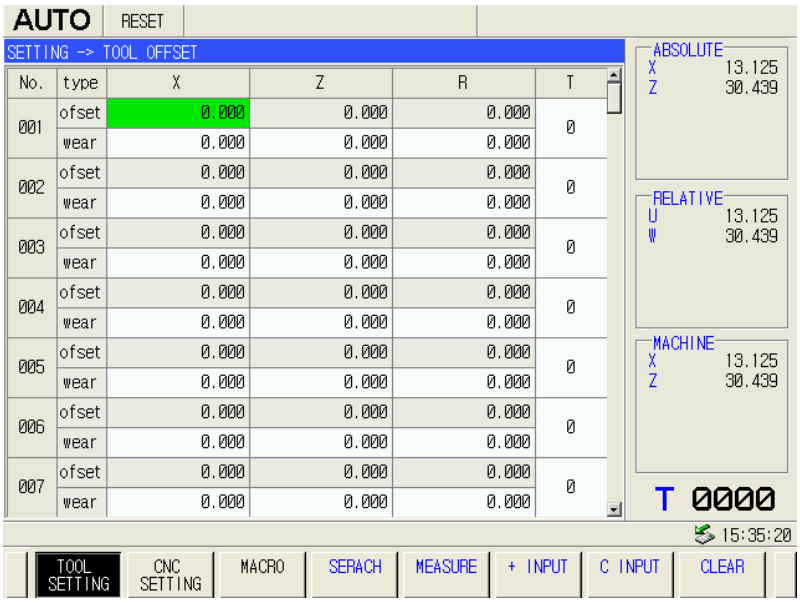


Fig.7-1

（2）On the window, press or to select the window, and press or to select the tool offset number to be rewritten, and press or to select the axial offset Value, the wearing Value or T value of the assumed tool nose direction to be rewritten, which is shown as X axis offset of #001 tool offset in the above figure; About the relative relation of the assumed tool nose, refer to the tool nose radius compensation in the 4th chapter in *programming introduction*.

（3）Directly rewrite the tool offset Value, the wearing Value or the relative assumed tool nose direction number T through the numerical keys or the backspace key; or press

to make the selected tool offset value be input, such as X axis offset of #001 tool offset shown as the following figure, and then rewrite the tool offset Value, the wearing Value or the corresponding assumed tool nose direction number T through pressing the numerical keys or the backspace key.

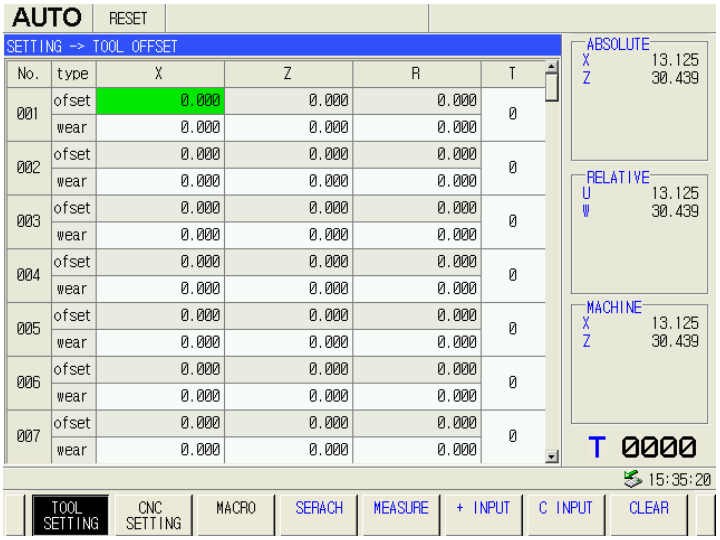


Fig.7-2

（4）Press INPUT to complete the input or rewriting, or switch into the other window to complete the rewriting.

（5）Move the cursor to set the other tool offset value, wearing value or T value of the assumed tool nose direction.

Note: The maximum value of the tool wearing compensation value can be rewritten through parameter 5013.

7.1.2 Measuring mode

（1）On the setting tool offset window, press TOOL OFFSET to access the tool offset management window;

（2）Press or to select the window, and press or to select the tool offset number to be rewritten, or press or to select the axial tool offset Value or the wearing value to be rewritten.

（3）Press MEASURE to access the measuring window to measure the tool offset value, which is shown as below:

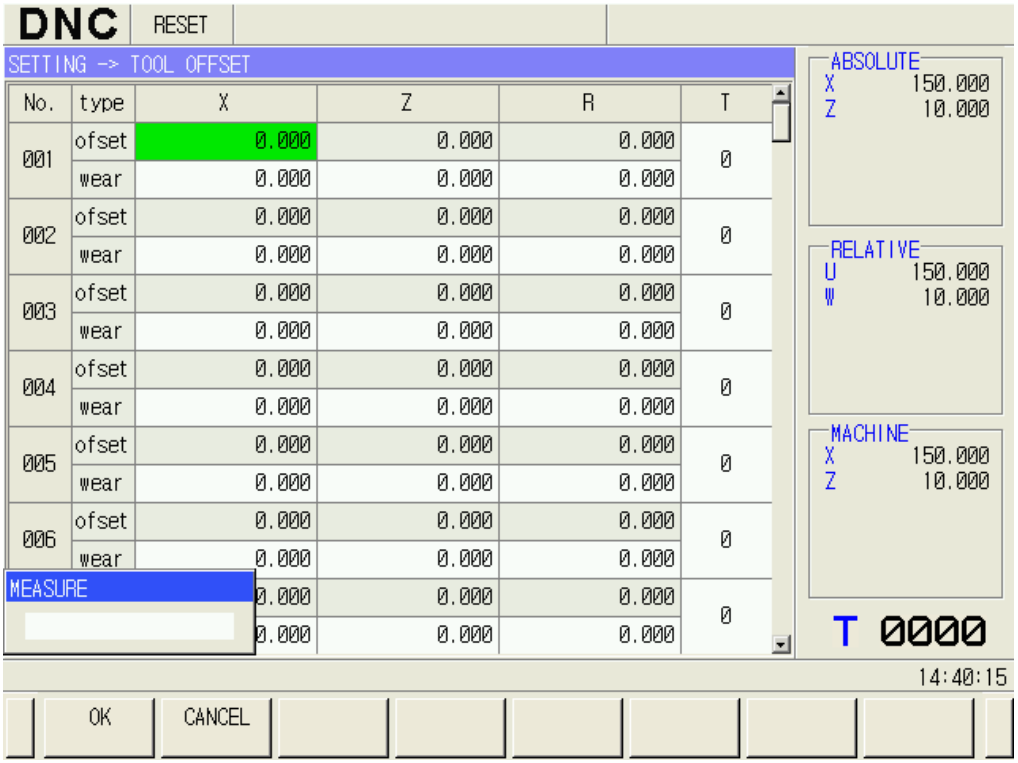
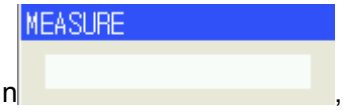
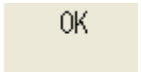


Fig.7-3

（4）Input “the coordinate axis number + axis value” to be measured in 

, press 

or

for in-position measuring;

（5）Calculating the offset value:

If the cursor is in the tool offset box, the tool wearing value is cleared, the tool offset value = the relative coordinate value – the input coordinate value;

If the cursor is in the tool wearing box, the tool wearing value remains unchanged, the tool offset value = the relative coordinate value — the input coordinate value — the wearing value relative to the coordinate axis.

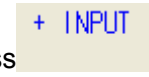
Note: The lathe tool-setting isn’t with the tool compensation value.

7.1.3 +input mode

（1）On the setting tool offset window, press

to access the tool offset management window;

（2）Press
or
to select the window, and press
or
to select the tool offset number to be rewritten, and press
or
to select the axial tool offset Value or the wearing value to be rewritten;

（3）Press 

, the selected tool offset value or the wearing value adds up one input value, which is shown as below:

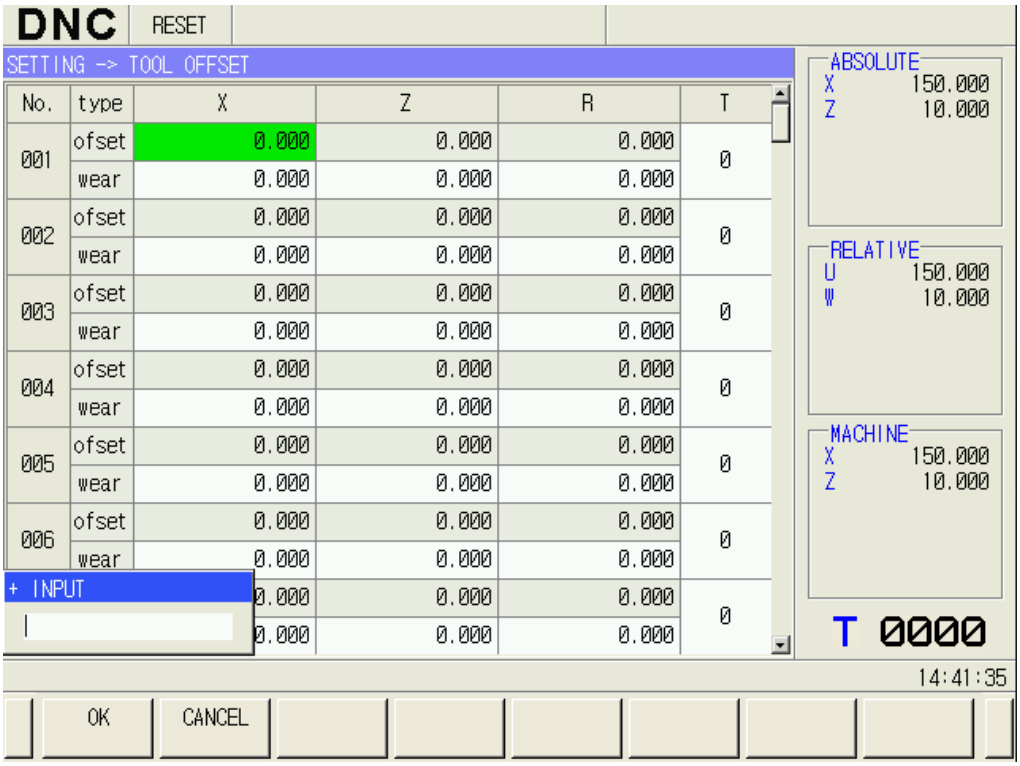
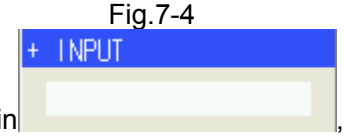


Fig.7-4

（4）Input one numerical value in 

, the value can be negative. Press OK or INPUT to complete the input;

（5）Calculating the offset value: The offset value or the wearing value = the original offset value or the original wearing value + the input numerical value.

7.1.4 C input method

（1）On the setting tool offset window, press

to access the tool offset management window;

（2）Press or to select the window, and press or to select the tool offset number to be rewritten, and press or to select the axial tool offset Value or the wearing value to be rewritten;

（4）Press C INPUT to access C input window, which is shown as below:

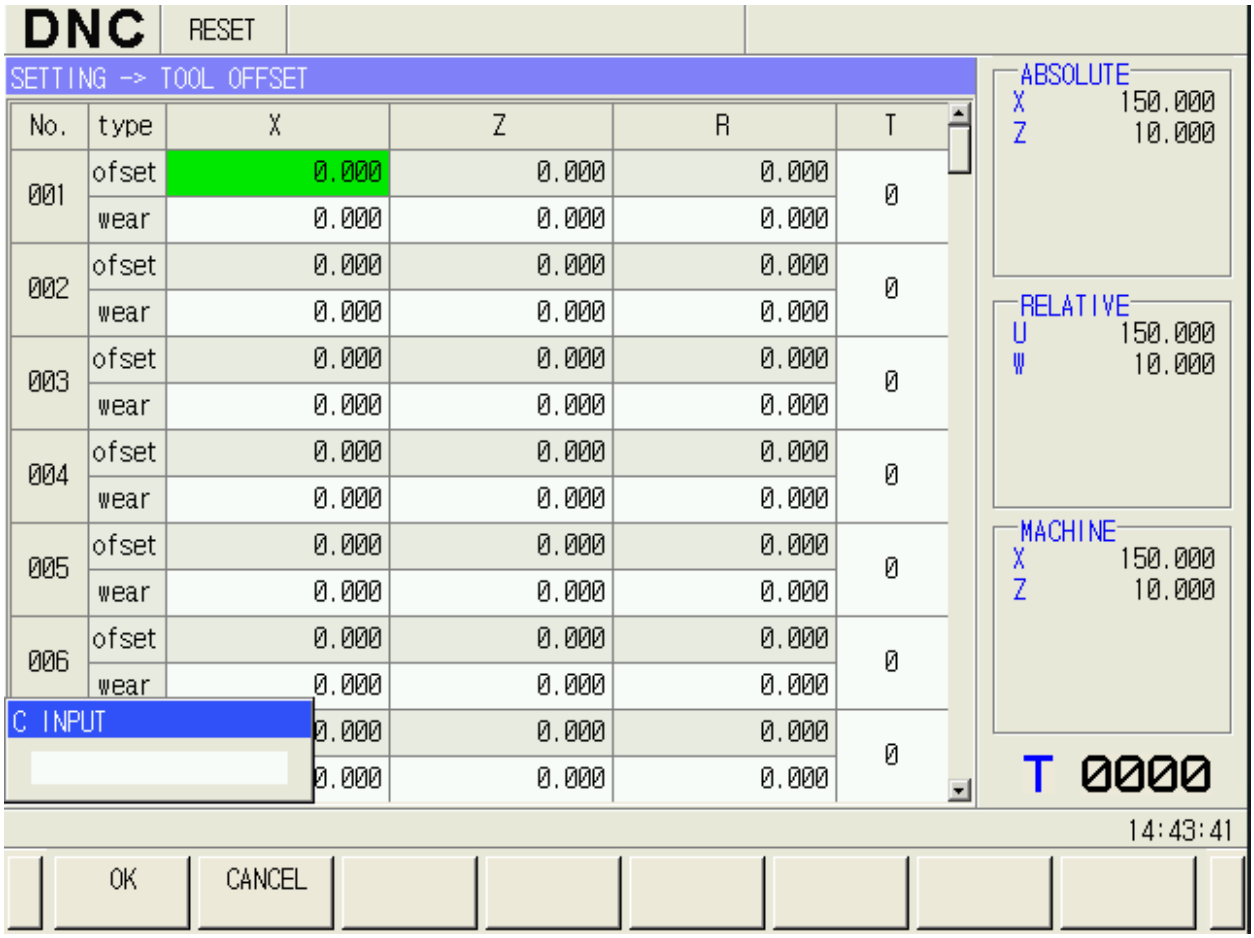
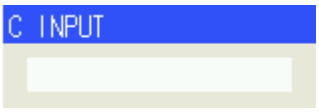


Fig.7-5

（5）Input the coordinate axis name to be measured in 

, press

for in-position measuring;

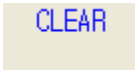
（6）Then, calculate the offset value;

Press C input button to input the axial number.

If the cursor is on the tool offset box, the tool wearing value remains unchanged, write in the tool offset value = the relative coordinate value – the tool wearing value;

If the cursor is on the tool wearing box, the tool offset value remains unchanged, write in the tool wearing value = the relative coordinate value – the tool offset value.

7.1.5 Clearing the offset value or the wearing value

On the tool offset management window, press or to select the window, and press or to select the tool offset number to be rewritten, and press or to select the tool offset Value, the wearing value or the tool number to be cleared; press 

to clear the selected tool offset value, the wearing value or the assumed tool nose direction number relative to the axis.

7.2 Fixed-Point Tool Setting

Fixed-point tool-setting is to set the tool offset Value through C input mode.

The steps are as below:

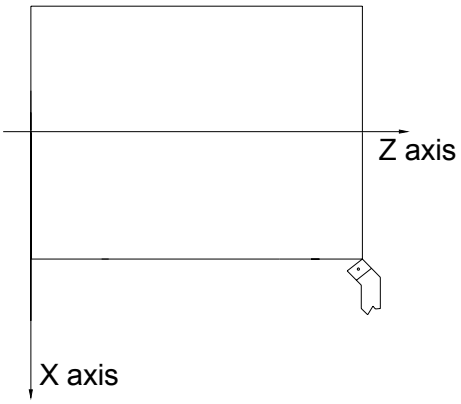


Fig. 7-6

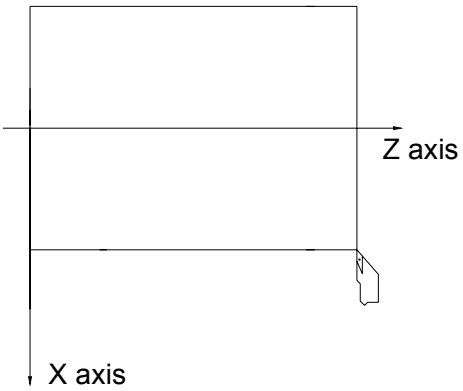


Fig. 7-7

1. Firstly confirm the tool compensation value in X or Z direction is 0; if not, the tool compensation values of all tool numbers must be cleared;

2. The tool offset number is 00 (such as T0100 and T0300);

3. Select any tool (normally the first tool during processing is taken as the datum tool)

4. The nose of the datum tool positions in some point (tool-setting point), which is shown as figure A;

5. In MDI mode, G50 X__ Z__ command, on the program window, sets the work piece coordinate system;

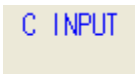
6. The value of relative coordinate (U, W) is cleared;

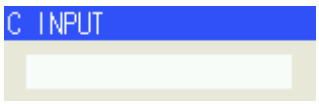
7. The tool traverses to the safe position, and the other tool is selected and traverses to the tool-setting point, which is shown as figure B;

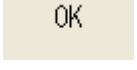
8. On the setting window, press

to access the tool offset management window, press

to select the tool offset number, or press or to select the window, and press or to select the tool offset or wearing Value to be rewritten;

9. Press 

to access C input window, input axial name 

, press 

and the tool offset value or the wearing value is set in the corresponding offset number;

10.Repeat the steps of 7～9, other tools can be set.

7.3 Trial Cut Toolsetting

After the coordinate system is set, the trial cutting tool is to set the tool offset value through measuring input method.

The steps are as below: (set the work piece coordinate system based on the work piece face):

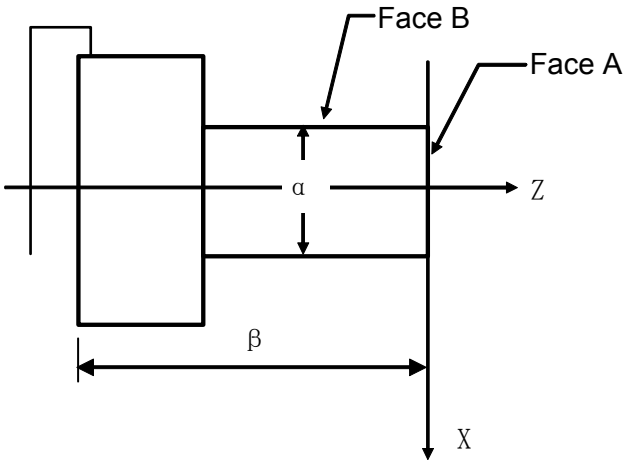


Fig.7-8

1. Ensure each axis on the machine has executed the machine zero return;
2. Any tool is selected and the tool offset number is 00(such as T0100, T0300);
3. The tool cuts along face A;
4. When Z axis remains still, the tool retracts along X axis and the spindle stops revolving;
5. On the setting window, press TOOL OFFSET to access the tool offset management window, press SERACH to select the tool offset number, or press ▤ or ▤ to select the window, and press ⇧ or ⇩ to select the tool offset or the wearing Value to be rewritten;
6. Press MEASURE to access the measuring window, input Z and measuring value β in MEASURE _____, and then press OK, Z axis tool offset value or its wearing value is set in the corresponding offset number;
7. The tool cuts along surface B;
8. When X axis remains still, the tool retracts along Z axis and the spindle stops revolving;
9. Measure diameter “α”;
10. Press MEASURE to access the measuring window, input X and the measuring value α in MEASURE _____; After pressing OK, X axis tool offset value or its wearing value is set in the corresponding offset number;
11. The tool traverses to the safe position for changing into the other one;

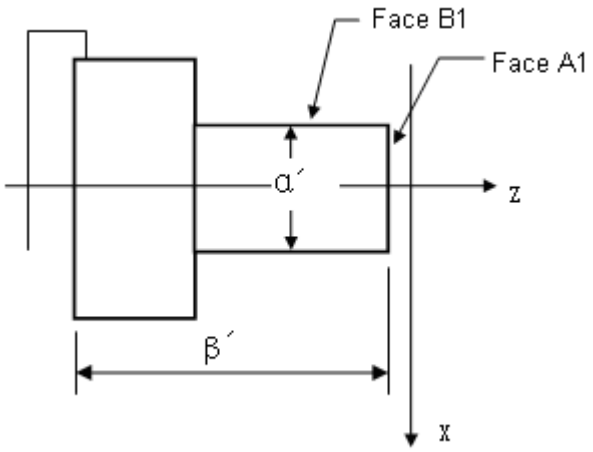


Fig.7-9

12. The tool cuts along face A1 as Fig.7-9;

13. When Z axis remains still, the tool retracts along X axis and the spindle stops rotation; measure the distance “β” from face A1 to the origin of the work piece coordinate system;

14.On the setting window, press TOOL OFFSET to access the tool offset management window, and press SERACH to select the corresponding tool offset number, and press ⇧ or ⇩ to select the window, and press ⇧ or ⇩ to select the tool offset or the wearing value to be rewritten;

15. Press MEASURE to access the measuring window, input Z and measuring value β ´ in MEASURE; After pressing OK, Z axis tool offset value or its wearing value is set in the corresponding offset number;

16. The tool cuts along face B1;

17. When X axis remains still, the tool retracts along Z axis and the spindle stops revolving; 18. Measure distance “α ´ ”;

19. Press MEASURE to access the measuring window, input X and the measuring value α ´ in MEASURE, and then press OK and X axis tool offset value or its wearing value is set in the corresponding offset number;

20.About the method of setting other tools, repeat the steps of 10～19.

Measuring method is to set the differential value between the tool reference position (such as the tool nose position) and the actual tool nose position during processing as the tool offset value. For example: when the coordinate value of face B is 50.0, the actual measured value is α=49.0, then the tool offset value in X direction is 1.0.

Note: After the machine zero return toolsetting is executed, G50 cannot be used to set a workpiece coordinate system.

7.4 Position Record

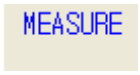
The position record key on the machine panel is valid when the parameter PRC (No.5005#2) is set to 1.

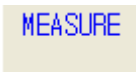
Position record toolsetting operation mode:

1）Cut the outer or end face in Manual mode.

2）When

on the operation panel, the workpiece coordinate values of X (X axis of three basic axes) and Z (Z axis of basic axes) have been recorded to CNC.

3）Then, the tool retraction and the spindle stop are executed. When the outer direction is executed, the diameter is measured, 

is pressed to input X+ the measured value for the tool compensation number corresponding to the tool offset, and so the operation is completed; when the end face being executed, the length between it and the datum level is measured (the datum level: Z=0), 

is pressed to input X+ the measured value for the tool compensation number corresponding to the tool offset, and so the operation is completed.

Note 1: If

is pressed many time, the coordinate position when the

pressed last is only recorded by the CNC.

Note 2: In the point-to-point toolsetting mode, the position record is used: after the reference tool is set and other tools reach the toolsetting points,

is pressed, i.e. the current coordinate position can be recorded, the tool offset is input based on the fixed-point toolsetting method after the tool retraction is executed.

7.5 Automatic Tool Compensation

When an automatic toolsetting device is installed on the machine, the CNC sends commands used to the automatic measure, and automatically measures or determines the compensation amount of the tool. Firstly, the CNC sends a command used to the measure, and the tool traverses to the measure position. The CNC automatically measures the coordinate difference between the measure point and the commanded measured position, which is taken as the tool compensation amount. When the tool has compensated, it traverses to the measure position. The coordinate differences between the measure point and the commanded are summed to the current compensation amount which is set.

Note: When the automatic tool compensation function is used, IGA (NO.6140#7) is set to "0".

Automatic measure command:
X axis:G36
Z axis:G37

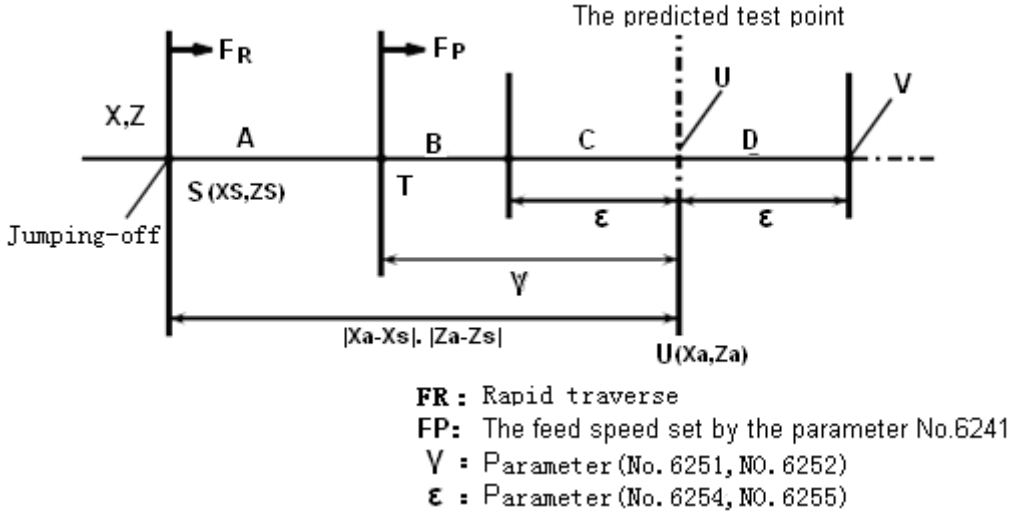
Measured position arrival signal:
XAE(X3.6) corresponds to G36
ZAE(X3.7) corresponds to G37

G36, G37 automatic tool offset use

In Fig.7-10, one of G36, G37 is commanded to the block, and the tool firstly traverses to the

commanded measure position at the rapid traverse mode. And, the tool decelerates to stop at the position which is Y from the measured position, and then, traverses to the measure position at the measure speed set by No.6241～6243.

And then, after the tool approaches the distance ε, and when the measure position arrival signal corresponding to the command is “1” in the course of the overtravel distance ε, the above compensation amount is updated, and traverse command of the block ends. When the measure position arrival signal has not become “1” from the measure position to the overtravel distance ε,



FR : Rapid traverse
FP: The feed speed set by the parameter No.6241
γ : Parameter(No.6251,NO.6252)
ε : Parameter(No.6254,NO.6255)

Fig. 7-10

Note 1: Refer *GSK988T Programming User Manual* about G36, G37;
Note 2: Refer to the user manual supplied by the machine manufacturer about the automatic toolsetting device;
Note 3: No.6241: set X feedrate when automatic tool compensation;
No.6251: set Y value of X axis when automatic tool compensation;
No.6254: set ε value of X axis when automatic tool compensation;
No.6242: set feedrate of Z axis when automatic tool compensation;
No.6252: set γ value of Y axis when automatic tool compensation;
No.6255: set ε value of Z axis when automatic tool compensation;

Chapter Ⅷ Setting and Display Graphs

8.1 Setting the Graph Parameter

Before display the path, the relative message of the path display or the graphic simulation must be set.

The graph message mainly sets the offset value of each coordinate axis, the length and the diameter of the processing work piece, the magnification ratio of the graph path and that of the graph simulation. The detailed steps are as below:

（1）Press GRAPH to access the graph window;

（2）On the graph window, press GRAPHSET to access the setting graph parameter window and it is shown as below:

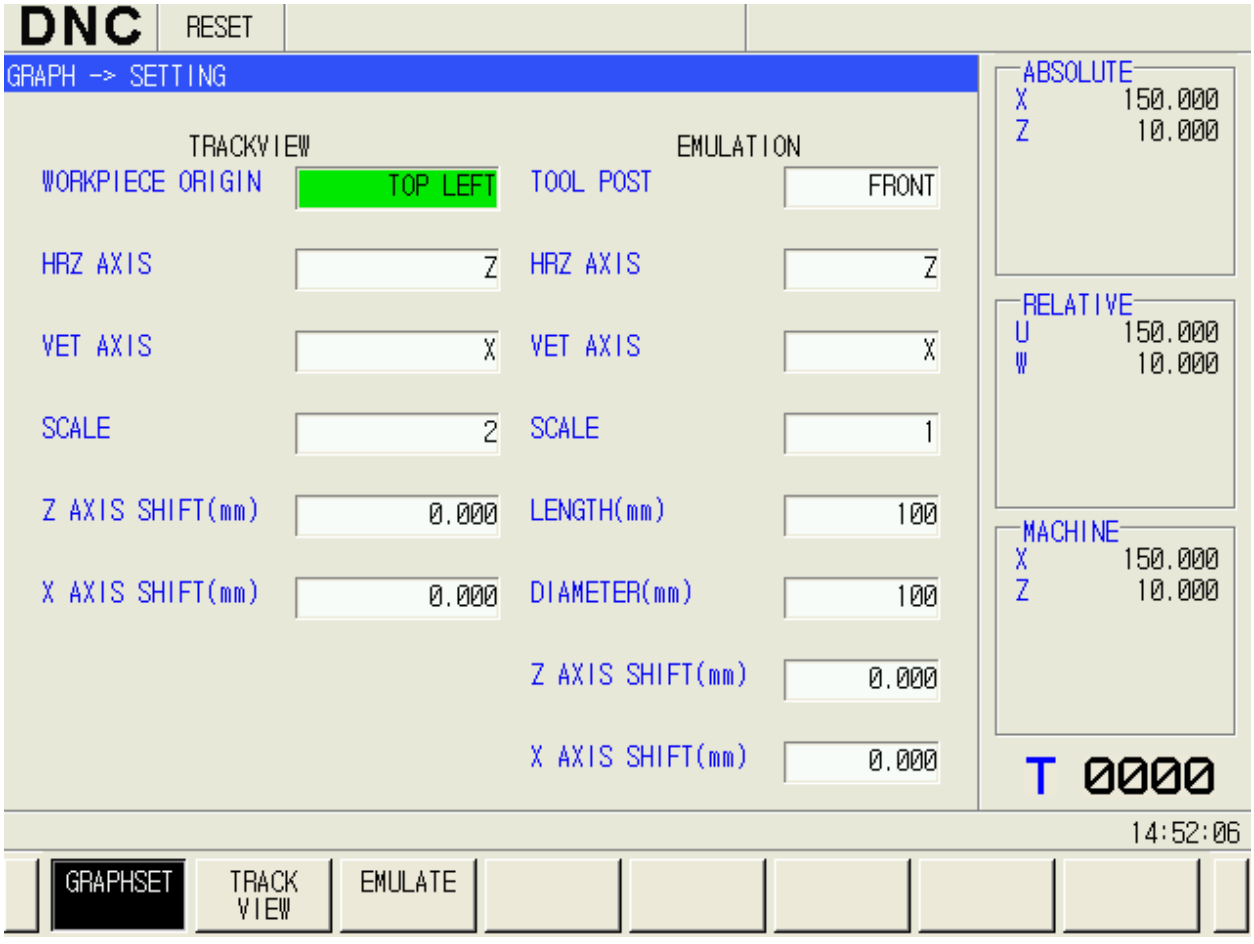


Fig.8-1

（3）Press ⇧ or ⇩ to select the item to be set, such as the cross axis shown as below:

（4）Press INPUT and the selected item can be input, such as the cross axis shown as below:

（5） Press ⇧ or ⇩ to select the item to be set and press INPUT to confirm the rewriting is completed.

（6）Repeat the above operation to set the other parameter.

Note 1: The setting on the window is only for the path display and the display on the graph simulation window.
Note 2: The path display and the graph simulation are executed based on the machine coordinate. When the path and the graph are not displayed, please modify the coordinate axis offset.
Note 3: The horizontal axis must be Z axis or the aixs which is parallel with Z.

8.2 Path Graph Display and Operation

Through the graph path display, real-time check the path which the tool traverses.

（1）Press GRAPH to access the graph window;

（2）On the graph window, press TRACK VIEW to access the path window, display the program path which is being executed and it is shown as below:

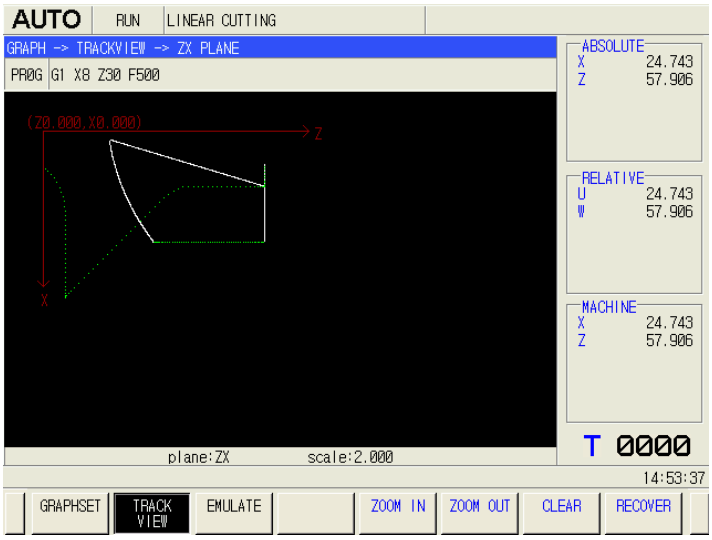


Fig.8-2

At the bottom of the path screen in the figure, it displays the coordinate level of the current path and the magnification ratio of the path graph. On the top of the figure, it displays the running mode and the state of the current system. On the right of the screen, it displays the current abosolute coordinate value, the relative coordinate value and the mode command.

The path graph can be operated as below:

（1）Press ZOOM IN or ZOOM OUT, the path graph can be zoomed out or in, and the previous ones can be cleared.

（2）Press CLEAR to clear the screen path.

（3）Press RECOVER and the path graph can be restored as the original normal position and the previous ones can be cleared.

（4）Repetitively press ⇧, ⇩, ⇦ or ⇦ to move the path graph up, down, left or right.

Note 1: The name for each axis can be set by parameter #1020, and the name can be set as different letters, and then, at the bottom of the path window, name of each coordinate level and that of the path coordinate change correspondingly.

Note 2: The system has only fron-rear tool post without left-right tool post, and the horizontal axis is set to only Z axis.

8.3 Simulation graph display and operation

Through the graph simulation, real-time check the complete cutting process of the part.

（1）Press GRAPH to access the graph window;

（2）On the graph window, press EMULATE to access the simulation graph window and it is shown as below:

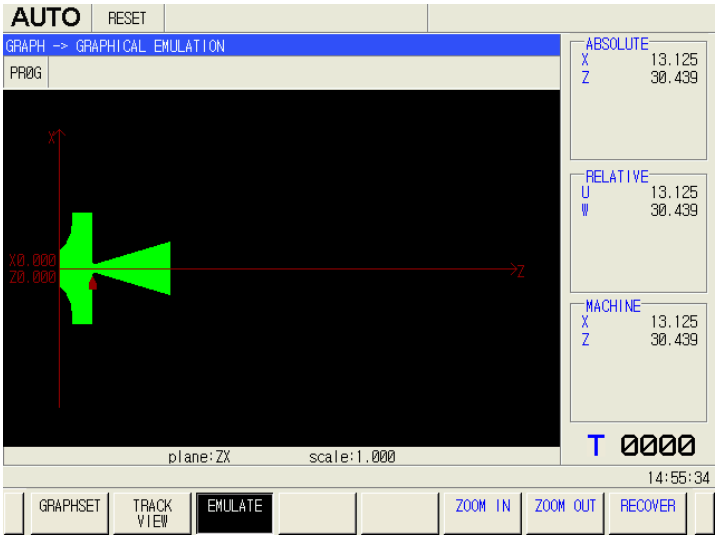


Fig.8-3

On the top of the figure, it displays the running mode and the state of the current system; on the right of the screen, it displays the message of the current absolute coordinate value, the relative coordinate value and the current tool number, etc.

In the figure, it only displays the simulation graph message of XZ coordinate level; at the bottom of the graph simulation screen, it displays the coordinate level which the current simulation graph is, and the magnification ratio of the simulation graph.

During the graph simulation process, the simulation graph can be operated as below:

（1）Press ZOOM IN or ZOOM OUT, the simulation graph can be zoomed in or out, and the previous simulation graph message can be cleared;

（2）Press RECOVER and the simulation graph can be restored as the original size and position, and the previous simulation graph message can be cleared.

（3）Respectively press , , or , and the simulation graph can move up, down, left or right.

Note: The name for each axis is set by parameter #1020, and each axis name can be set as the different letters, and then, at the bottom of the path window, the coordinate level and the path coordinate names can change correspondingly.

Chapter Ⅸ U disk Use

9.1 Sending a Program

Create a file in the U disk root catalog, and the file is saved as NCPROG. The program required to send is copied to the file.

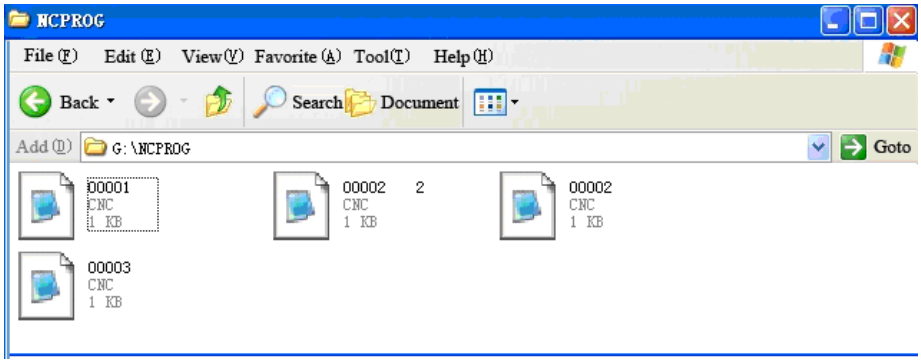


Fig. 9-1

After the above operations are executed, the U disk is inserted into the USB interface. When occurs in the bottom right corner, it means the U disk is connected, at the moment, PROGRAM is pressed, USB is pressed to enter the U disk file directory. As shown in Fig.9-2, press ⇧, ⇩ to select the required program to copy, press OUTPUT, i.e. the selected program in the U disk is copied to the local directory.

For example: For copying O0001 program in the U disk to the local direction, the user firstly uses the cursor to select the program in the U disk, and presses >, presses OUTPUT, and so the program is copied to the local directory.

The program in the local directory is copied to the U disk as long as the previous steps are executed in the local directory window.

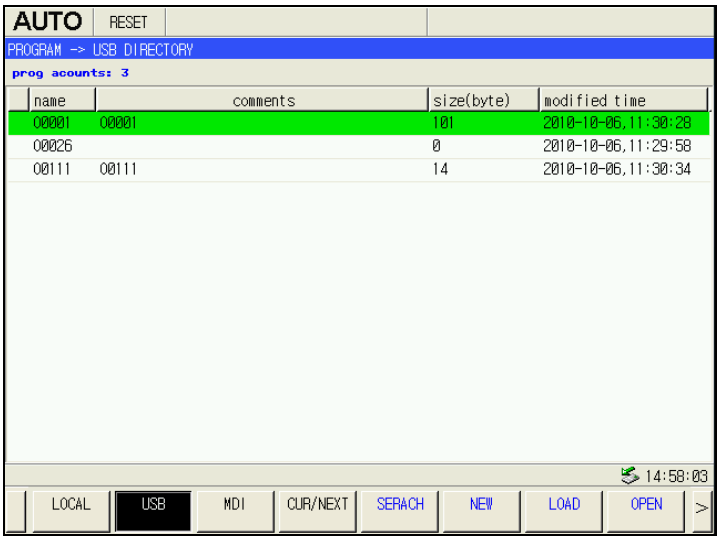


Fig. 9-2

9.2 Backup Value

GSK988T system can backup the system files and parameters to U disc to recover them later.

9.2.1 System file backup

Backup parameters, tool offset, pitch compensation, tool life, macro variable and other Value by the U disk, which is convenient to recover when the mistaken operations cause the mistaken Value. The operation steps are as follows:

1. Insert the U disk, and the system confirms it has read the disk;
2. Press SYSTEM to enter the system window, press MEMORY DEVICE to enter the file management display window as shown in Fig.9.2:

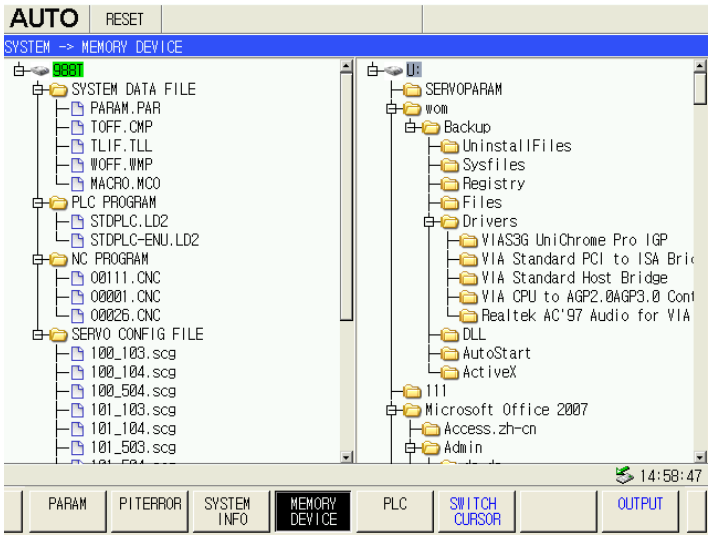


Fig. 9-3

3. There are five sub-files of the system file in the window: PARAM.PAR parameter, TOFF.CMP tool offset, TLIF.TLL tool life, WOFF.WMP pitch compensation, MACRO.MCO macro variable. Move up/down to the file to backup, press INPUT to select the file. When the cursor selects the file, at the moment, INPUT is pressed to select all files in the folder, as shown in Fig.9-3.
4. OUTPUT is pressed after the file is selected, and when 'Select Output Path' is popped, OK is pressed to copy the file after the selection has been done. Pull out the U disk after the file copy has been completed.
5. When the backup is needed to recover, the U disk is inserted, SWITCH CURSOR is pressed to switch the cursor the U disk, the cursor is moved to find the file to recover the Value, INPUT is selected, and INPUT is pressed to directly recover the file in the local directory. The system is turned on again after the file is recovered, otherwise, maybe some Value is invalid.

Note 1: When the parameters are recovered, their switches must be opened, and some parameters cannot recover because of authority.

Note 2: Recovering the pitch compensation can be done with the authority more than Level 2.

9.2.2 Servo parameter backup

9.2.2.1 Exporting servo parameters

1. Insert the U disk, and confirm the system has read it.

2. Press SYSTEM to enter the system page, press PARAM, and then SERVO PARAM to enter the servo parameter management page. The page is shown in Fig. 9-4:

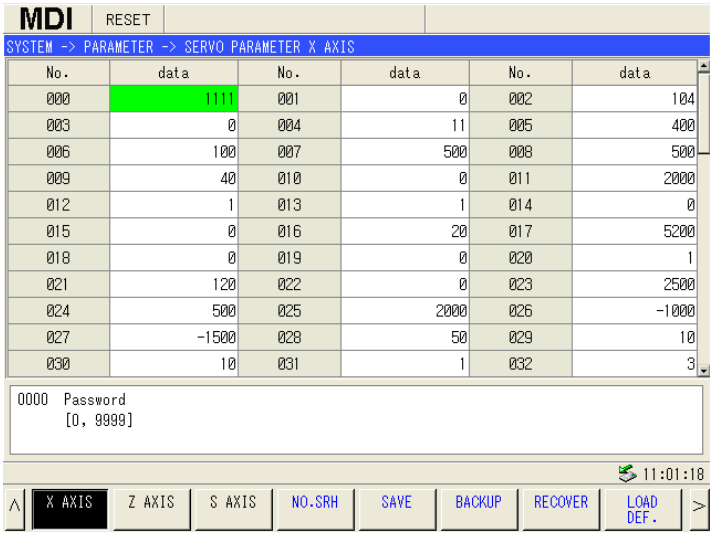


Fig. 9-4

3. As the above figure, select X-axis servo parameter, press >, then EXPORT PARAM. As the following Fig. 9-5, because the previous selection is X-aixs, the exported file name is changed into X, OK is pressed, i.e., X-axis parameter file is backuped to the U disk.

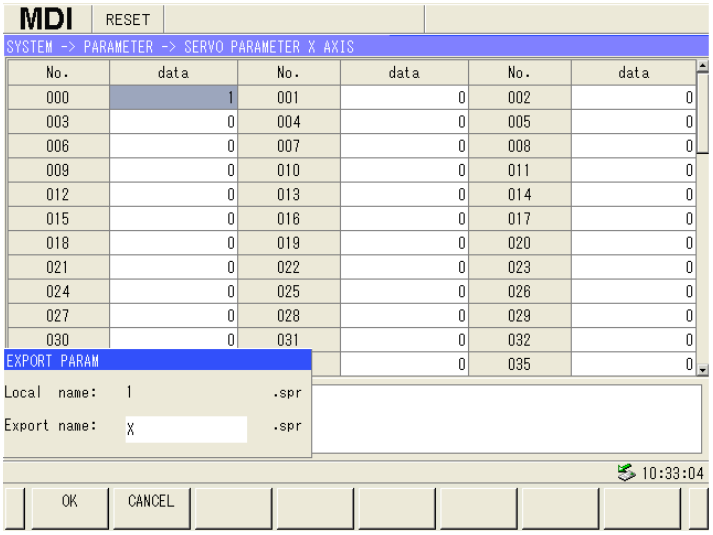


Fig. 9-5

4. Select other axes, repeat the above operations, i.e., the parameter files of other axes are backuped to the U disk.

5. After the Step 3 is done, the file “SERVOPARAM” is created in the U disk, the previous backuped parameter files are saved in the file as Fig. 9-6:

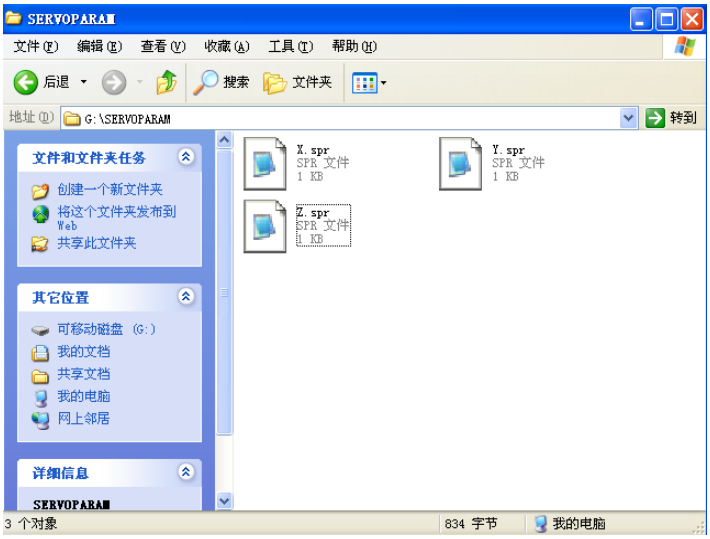


Fig. 9-6

9.2.2.2 Importing servo parameters

1. Ensure that the file“SERVOPARAM” has been created in the U disk, and the backuped servo parameters are saved to the file as Fig. 9-6.
2. Insert the U disk, and confirm the system has read the U disk.
3. In the servo parameter management page, select the axis which parameters will be imported. For X-axis, press [>], then [IMPORT PARAM]伺 and a dialog box pops up to select the correct parameter file, like X-aixs in Fig. 9-7, press [OK] and the parameters in the U disk are imported to the system.

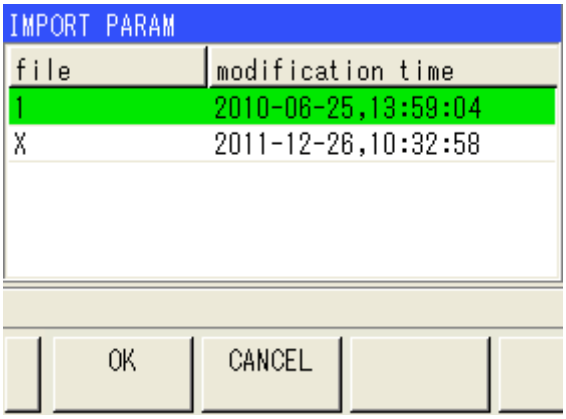


Fig. 9-7

4. After the importing is succeeded, the valid parameter is selected as Fig. 9-8. press [SELCT EFF. PAR], then press [IMORT PARAM] to use the imported parameter, a dialoge box pops up as Fig. 9-8, press [OK] to complete the operation, or press [CANCEL] to cancel it.

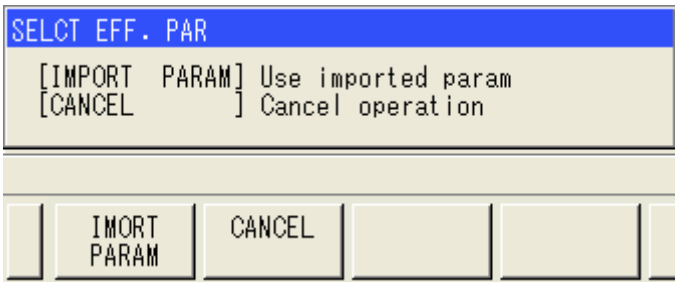


Fig. 9-8

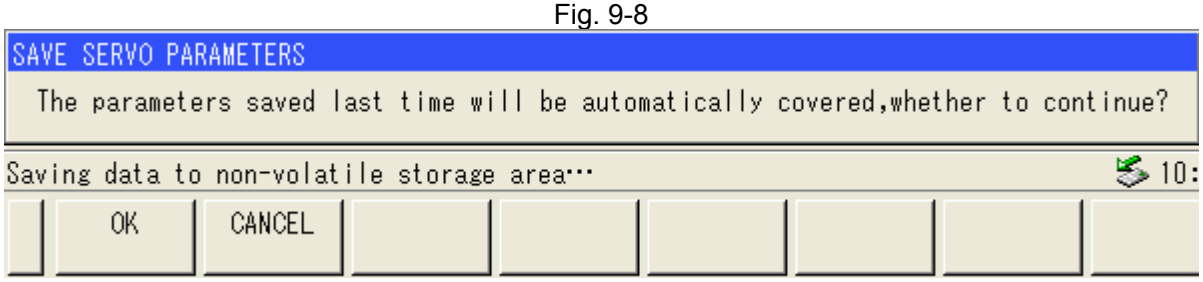


Fig. 9-9

5. Repeat the previous operations, i.e., the parameter files of other axes can be imported to the system.

Note 1: After servo parameters of all axes are imported, the machine must be turned on again to use.
Note 2: Exporting and importing can be done in MDI mode with the authority more than Level 3.

Chapter Ⅹ Processing Examples

10.1 Outer End Face Machining

1）Machining the workpiece is shown in Fig. 10-1 and the rod isΦ50×100mm

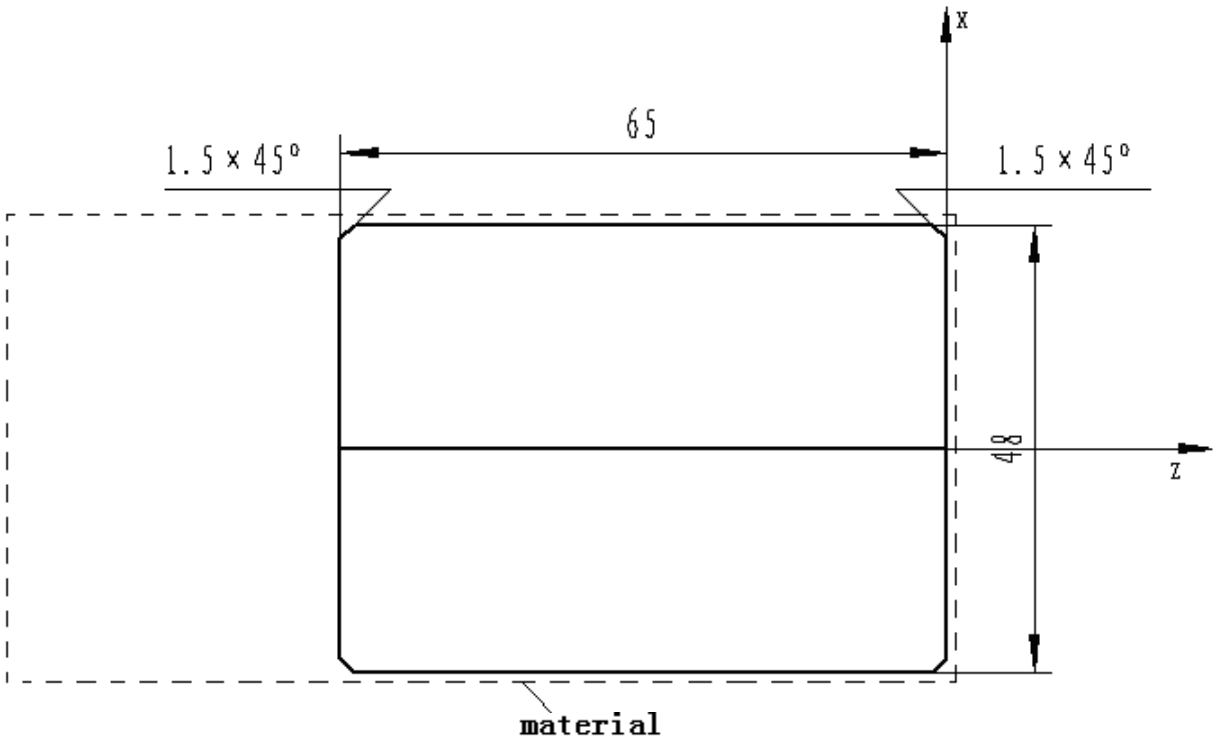
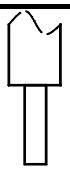


Fig. 10-1

2）2 tools used to machine below:

Tool No.	Tool type	Remark
No. 1 tool		Outer tool
No.2 tool		Cutting tool with the tool width 3mm

3）Editing a program

According to the mechanical processing and introduction of the commands in the manual, set the work piece coordinate system shown as Fig 10-1; edit the programs shown as below:

O 0 0 0 1 ;		Program name
N 0 0 0 0	G0 X150 Z50;	Position to the safe place to change the tool
N 0 0 0 5	M12;	Clamp the chuck
N 0 0 1 0	M3 S800;	The spindle is on, and its speed is 800
N 0 0 2 0	M8;	The cooling is ON

N0030	T0101;	Change into the 1st tool
N0040	G0 X136 Z2;	Close to the work piece
N0050	G71 U0.5 R0.5 F200;	Cutting depth is 1mm, the tool retracts for 1mm.
N0055	G71 P0060 Q0150 U0.25 W0.5;	X axis leaves for 0.5mm, 0.5mm surplus in Z axis
N0060	G0 X16;	Close to the work piece face
N0070	G1 Z-23;	TurningΦ16 outer circle
N0080	X39.98;	Turning face
N0090	W-33;	TurningΦ39.98 outer circle
N0100	X40;	Turning face
N0105	W-30;	Turning Φ40 outer circle
N0110	G3 X80 W-20 R20;	Turning convexo arc
N0120	G2 X120 W-20 R20;	Turning concave arc
N0130	G1 W-20;	Turning Φ120 outer circle
N0140	G1 X130 W-5;	Taper turning angle
N0150	G1 W-25;	TurningΦ130 outer circle
N0160	G0 X150 Z185;	Return to the tool change point after roughing
N0170	T0202;	Change into #2 tool, execute #2 tool offset
N0180	G70 P0060 Q0150;	Finishing cycle
N0190	G0 X150 Z185;	Return to the tool change point after roughing
N0200	T0303;	Change into #3 tool, execute #3 tool offset
N0210	G0 Z-56 X42;	Close to the work piece
N0220	G1 X30 F100;	Grooving Φ30
N0230	G1 X37 F300;	Return
N0240	G1 X40 W1.5;	Chamfering
N0250	G0 X42 W30;	Leave the grooving tool width
N0260	G1 X40 ;	
N0262	G1 X37 W1.5;	Chamfering
N0264	G1 X10;	Grooving Φ10
N0266	G0 X17 Z-1;	
N0268	G1 X16;	
N0270	G1 X14 Z0 F200;	Chamfering
N0280	G0 X150 Z50;	Return to the tool change point
N0290	T0404 S100;	Change into #4 tool, set the spindle speed as 200 r/min.
N0300	G0 X42 Z-54;	Close to the work piece
N0310	G92 X39 W-34 F3;	Threading cycle
N0320	X38;	Feed 1mm and cut the 2nd time
N0330	X36.4;	Feed 0.6mm and cut the 3rd time
N0332	X36;	Feed 0.4mm and cut the 4th time

N 0 3 4 0	G0 X150 Z50;	Return to the tool change point
N 0 3 5 0	T0100;	Change into #1 tool
N 0 3 6 0	M5;	The spindle is OFF
N 0 3 7 0	M9;	The cooling is OFF
N 0 3 8 0	M13;	Release the chuck
N 0 3 9 0	M30;	End of a program

4）Toolsetting and run

(1) The tool traverses to the safe position; in MDI mode, the system executes T0100 and cancels the tool offset on the program window;

(2) The tool traverses and cuts along the work piece face as shown in Fig.10-2;

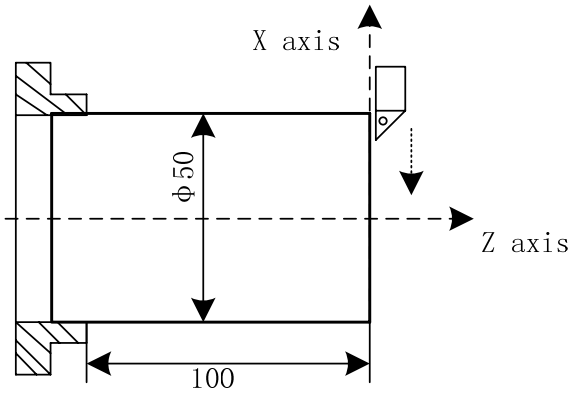


Fig.10-2

(3) When Z axis remains still, the tool is released along X axis, and the spindle rotation stops, the system is switched to the tool offset window, the cursor moves to No.001 offset, MEASURE is pressed and the system enters the measure input window, Z0 in the input window MEASURE is input, and OK is pressed, and so, Z offset value has been input;

(4) The tool traverses and cuts along the outer circle of the work piece as shown in Fig.10-3;

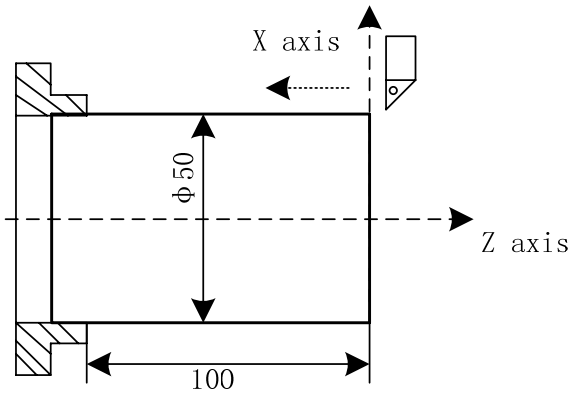


Fig.10-3

(5) When X axis remains still, the tool is released along Z axis, and the spindle rotation stops, and the outer dimension of the workpiece is measured (the measured value is 49.5mm); the system is

switched to the tool offset window, the cursor moves to No.001 offset, MEASURE is pressed and the system enters the measure input window, Z0 in the input window MEASURE is input, and OK is pressed, and so, Z offset value has been input;

(6) Traverse the tool to the safe position, press the tool change key to execute No.2 tool in Manual mode;

(7) Start the spindle and traverse the tool to the toolsetting point as shown in Fig.10-4, point A;

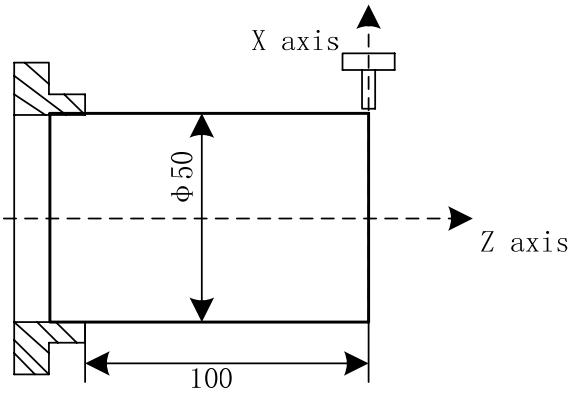


Fig.10-4

(8) Switch into the tool offset window, the cursor moves to #002 offset, press MEASURE to access the measuring window, and input X135 in MEASURE, and then press OK. Use the same method to input **Z0**;

（9）The toolsetting is completed and the tool traverses to the safe position;

(10) Press CYCLE START to automatically machine the workpiece in Auto mode;

(11) Modify the tool wear value to the tolerance range of the workpiece dimension when the measured workpiece dimension is different from the actual.

10.2 Compound Machining

Machining the workpiece is shown in Fig. 10-5 and the rod is Φ136×190mm

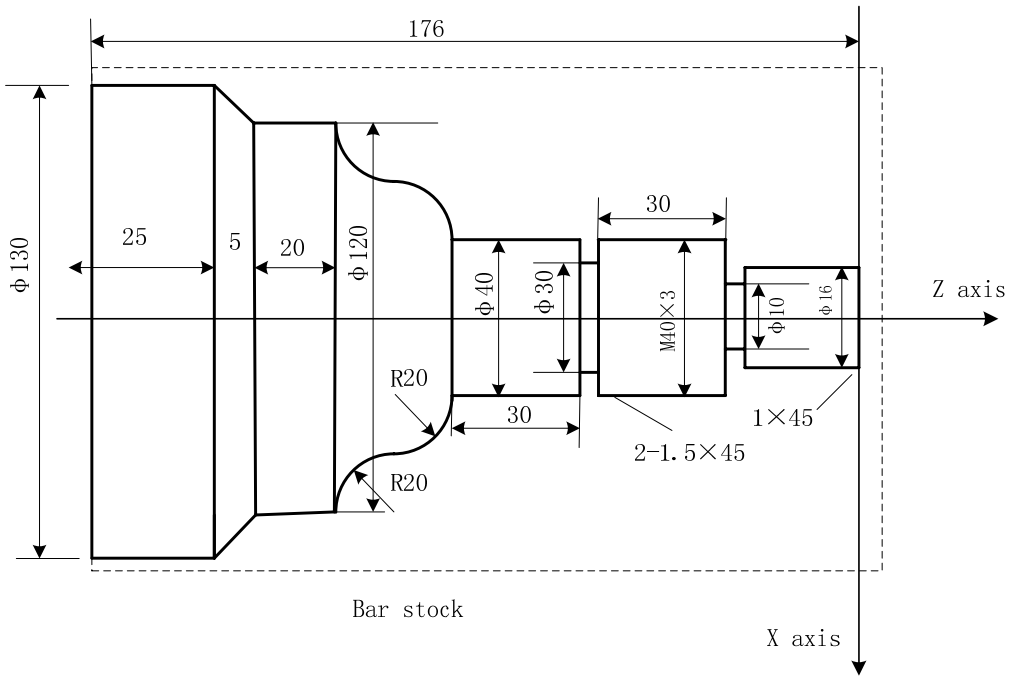
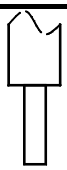


Fig.10-5

Tool No.	Tool type	Remark
#1 tool		Outer roughing tool
#2 tool		Outer finishing tool
#3 tool		Grooving tool, its width is 3mm
#4 tool		Thread turning tool, the nose angle is 60°

3）Editing a program

According to the mechanical processing and introduction of the commands in the manual, set the work piece coordinate system shown as Fig 12-1; edit the programs shown as below:

O 0 0 0 1 ;		Program name
N 0 0 0 0	G0 X150 Z50;	Position to the safe place to change the tool
N 0 0 0 5	M12;	Clamp the chuck
N 0 0 1 0	M3 S800;	The spindle is on, and its speed is 800
N 0 0 2 0	M8;	The cooling is on
N 0 0 3 0	T0101;	Change into the 1st tool
N 0 0 4 0	G0 X136 Z2;	Close to the work piece
N 0 0 5 0	G71 U0.5 R0.5 F200;	Cutting depth is 1mm, the tool retracts for 1mm.
N 0 0 5 5	G71 P0060 Q0150 U0.25 W0.5;	X axis leaves for 0.5mm, 0.5mm surplus in Z axis
N 0 0 6 0	G0 X16;	Close to the work piece face
N 0 0 7 0	G1 Z-23;	TurningΦ16 outer circle
N 0 0 8 0	X39.98;	Turning face
N 0 0 9 0	W-33;	TurningΦ39.98 outer circle
N 0 1 0 0	X40;	Turning face
N 0 1 0 5	W-30;	Turning Φ40 outer circle
N 0 1 1 0	G3 X80 W-20 R20;	Turning convexo arc
N 0 1 2 0	G2 X120 W-20 R20;	Turning concave arc
N 0 1 3 0	G1 W-20;	Turning Φ120 outer circle
N 0 1 4 0	G1 X130 W-5;	Taper turning angle
N 0 1 5 0	G1 W-25;	TurningΦ130 outer circle
N 0 1 6 0	G0 X150 Z185;	Return to the tool change point after roughing
N 0 1 7 0	T0202;	Change into #2 tool, execute #2 tool offset
N 0 1 8 0	G70 P0060 Q0150;	Finishing cycle
N 0 1 9 0	G0 X150 Z185;	Return to the tool change point after roughing
N 0 2 0 0	T0303;	Change into #3 tool, execute #3 tool offset
N 0 2 1 0	G0 Z-56 X42;	Close to the work piece
N 0 2 2 0	G1 X30 F100;	Grooving Φ30
N 0 2 3 0	G1 X37 F300;	Return
N 0 2 4 0	G1 X40 W1.5;	Chamfering
N 0 2 5 0	G0 X42 W30;	Leave the grooving tool width
N 0 2 6 0	G1 X40 ;	
N 0 2 6 2	G1 X37 W1.5;	Chamfering
N 0 2 6 4	G1 X10;	Grooving Φ10
N 0 2 6 6	G0 X17 Z-1;	
N 0 2 6 8	G1 X16;	

N 0 2 7 0	G1 X14 Z0 F200;	Chamfering
N 0 2 8 0	G0 X150 Z50;	Return to the tool change point
N 0 2 9 0	T0404 S100;	Change into #4 tool, set the spindle speed as 200 r/min.
N 0 3 0 0	G0 X42 Z-54;	Close to the work piece
N 0 3 1 0	G92 X39 W-34 F3;	Threading cycle
N 0 3 2 0	X38;	Feed 1mm and cut the 2nd time
N 0 3 3 0	X36.4;	Feed 0.6mm and cut the 3rd time
N 0 3 3 2	X36;	Feed 0.4mm and cut the 4th time
N 0 3 4 0	G0 X150 Z50;	Return to the tool change point
N 0 3 5 0	T0100;	Change into #1 tool
N 0 3 6 0	M5;	The spindle is off
N 0 3 7 0	M9;	The cooling is off
N 0 3 8 0	M13;	Release the chuck
N 0 3 9 0	M30;	End of a program

4）Toolsetting and run

(1) The tool traverses to the safe position; in MDI mode, the system executes T0100 and cancels the tool offset on the program window;

(2) The tool traverses and cuts along the work piece face as shown in Fig.10-6;

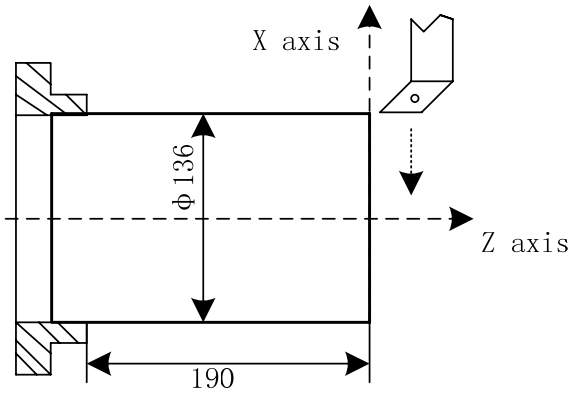


Fig.10-6

(3) When Z axis remains still, the tool is released along X axis, and the spindle rotation stops, the system is switched to the tool offset window, the cursor moves to No.001 offset, MEASURE is pressed and the system enters the measure input window, Z0 in the input window MEASURE is input, and OK is pressed, and so, Z offset value has been input;

(4) The tool traverses and cuts along the outer circle of the work piece as shown in Fig.10-7;

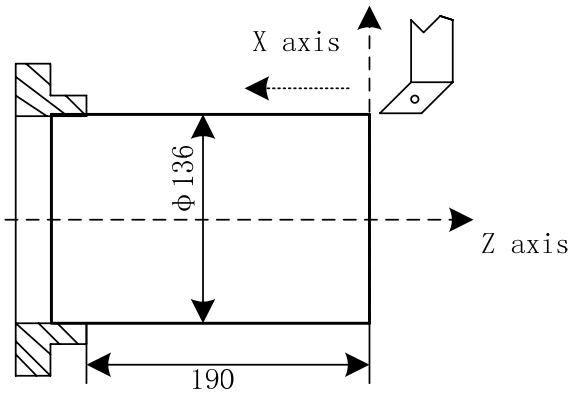


Fig.10-7

(5) When X axis remains still, the tool is released along Z axis, and the spindle rotation stops, and the outer dimension of the workpiece is measured (the measured value is 135mm); the system is switched to the tool offset window, the cursor moves to No.001 offset, MEASURE is pressed and the system enters the measure input window, X135 in the input window MEASURE is input, and OK is pressed, and so, X offset value has been input;

(6) Traverse the tool to the safe position, press the tool change key to execute No.2 tool in Manual mode;

(7) Start the spindle and traverse the tool to the toolsetting point as shown in Fig.10-8, point A;

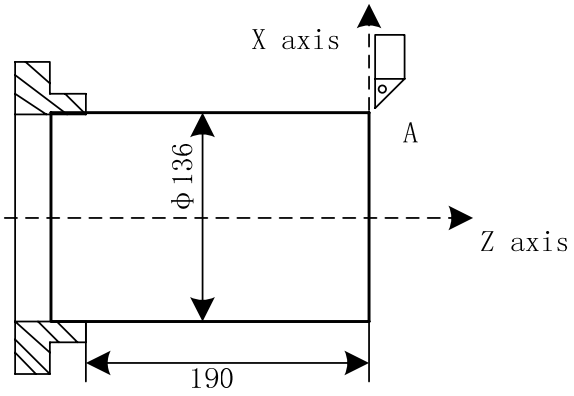


Fig.10-8

(8) Switch into the tool offset window, the cursor moves to #002 offset, press MEASURE to access the measuring window, and input X135 in MEASURE, and then press OK. Use the same method to input **Z0**;

（9）The tool traverses to the safe position, press the tool change key to execute No.3 tool in Manual mod;

(10) The spindle is started, the tool traverses to the toolsetting point as shown in Fig.10-9, point A;

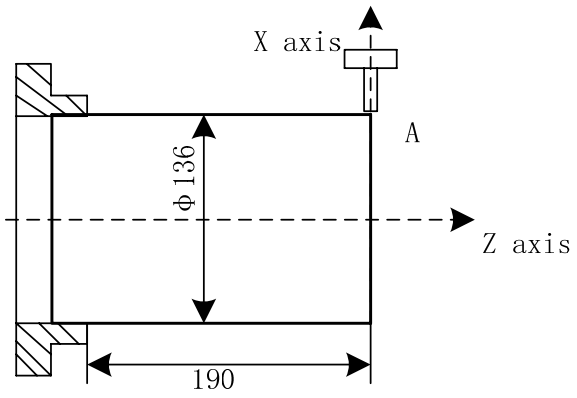


Fig.10-9

(11) The system is switched to the tool offset window, and the cursor moves to No.003 offset, X135, Z0 are input, and the input steps are the same those of the above (8);

(12) The tool traverses to the safe position, and the tool change key is pressed to execute the No.4 tool in Manual mode;

(13) The tool traverses to the toolsetting point as shown in Fig.10-10, point A;

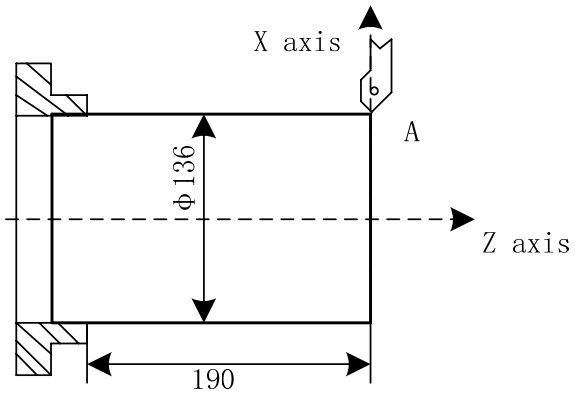


Fig. 10-10

(14) The system is switched to the tool offset window, and the cursor moves to No.004 offset, X135, Z0 are input, and the input steps are the same those of the above (8);

(15) The toolsetting is completed, and the tool traverses to the safe position;

(16) Press

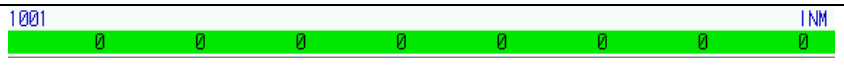
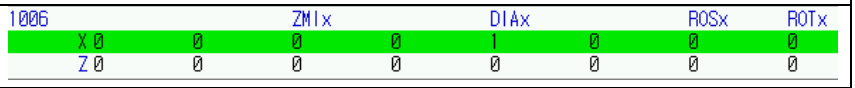
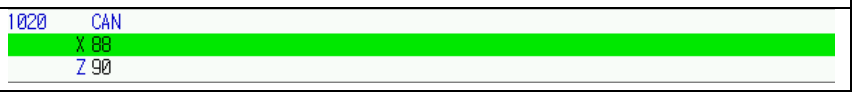
to automatically machine the workpiece in Auto mode;

(17) Modify the tool wear value to the tolerance range of the workpiece dimension when the measured workpiece dimension is different from the actual.

Chapter XI Parameters

This chapter mainly introduces CNC state and Value parameters through setting different parameters to realize the different requirements of function.

The parameter Value mainly includes the following six types:

Data type		Range
(1) Bit		8 bit 0 or 1
(2) Bit axis		
(3) Bit spindle		
(4) Word		-setting values are different according to different parameters. Please refer to parameters
(5) Word axis		
(6) Word spindle		

Value Types	Range
(1) Bit	8 digits, 0 or 1
(2) Bit axis	
(3) Word	-99 999 999～+99 999 999
(4) Word axis	

For the word axis parameter (3) and (4), the exact Value range is determined by specified parameters.

Each parameter should include the following information:

『Modification authority』: System authority (1st level), Machine authority (2nd level), Equipment management authority (3rd level), Operation authority (4th level), Limited authority (5th level)

『Parameter type』: bit, bit axis, bit spindle, word, word axis, word spindle

『Way of Validating』: Become valid immediately or after power-on

『Value Range』: In interval, by enumerating or special judgement)

『Default Setting』: 8 digits in binary system, or 32-digit integral value

Note 1: The 『Data Range』of bit type parameters is 0 or 1.
Note 2: When 『Validate method』 is not stated, the parameter will become valid immediately.
Note 3: When 『Parameter Type』 is not stated, the parameter is of bit type or word type.

(1) Bit (axis) type:

	#7	#6	#5	#4	#3	#2	#1	#0
0000								ABCx

『Modification authority』: System authority

『Way of Validating』: After power-on

『Default Setting』: 0000 0000

#0 ABCx The introduction of the parameter bit (axis) type is:

0: Allowed

1: Forbidden

(2) Word (axis) type:

1000	Parameter name

『Modification authority』: Equipment management authority

『Way of Validating』: After power-on

『Value Range』: 0～999

Explanation information of parameter in word (axis) type

11.1 Parameters Related to System Setting

	#7	#6	#5	#4	#3	#2	#1	#0
0000			SEQ			INI		

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#2 INI Input unit

0: Metric system

1: Inch system

#5 SEQ whether insert the sequence number automatically

0: No

1: Yes

Note: In EDIT or MDI mode, sequence number can be inserted automatically. The incremental value of sequence number is set in parameter.

11.2 Parameters Related to Interfaces of Input and Output

0123	Serial port baud rate (BPS)

『Modification authority』:Equipment management

『Value Range』: 4800, 9600, 19200, 38400, 57600, 115200

『Defualt Setting』: 115200

	#7	#6	#5	#4	#3	#2	#1	#0
0138		OWN						

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#6 OWN Overwrite confirmation displayed when nc data&prog is input&output:

0: Displayed

1: Not displayed

11.3 Parameters Related to Axis Control/Setting Unit

#7	#6	#5	#4	#3	#2	#1	#0

1001								INM

『Way of Validating』: After power-on

『Default Setting』: 0000 0000

#0 INM Least command unit on linear axis:

0: Metric system (metric machine)

1: Inch system (inch machine)

	#7	#6	#5	#4	#3	#2	#1	#0
1002					AZR		DLZ	

『Default Setting』: 0000 0000

#1 DLZ Function setting the reference positon without dog:

0: Disabled

1: Enabled(for all axes)

Note: When DLZ is 0, parameter 1005#1 (DLZx) can set valid/invalid for each axis.

#3 AZR G28 command without reference positon set causes:

0: Reference postion return with deleration dogs.

1: P/S alarm

Note: The function of reference point return without dog (when parameter 1002#1 (DLZ) is 1 or parameter 1005#1 (DLZx) is 1) is not related to the setting of AZR. If G28 is executed before reference point setting, P/S alarm is issued.

	#7	#6	#5	#4	#3	#2	#1	#0
1004		RPR					ISC	

『Way of Validating』: After power-on

『Default Setting』: 0000 0000

#1 ISC Least input increment & command increment

ISC	Least input unit, least command increment	Abbreviation
0	0.001mm, 0.001deg or 0.0001inch	IS-B
1	0.0001mm, 0.0001deg or 0.00001inch	IS-C

#6 RPR Least input increment of rotary axes tenfold of least command increment?

0: Not perform

1: Perform

	#7	#6	#5	#4	#3	#2	#1	#0
1005					HJZx		DLZx	ZRNx

『Parameter Type』: Bit axis

『Default Setting』: 0000 1000

#0 ZRNx Specify move command except for G28 without reference positon set causes (MEM, DNC or MDI).

0: Alarm

1: Not alarm

#1 DLZx Function for setting the reference positon without dogs.

0: Disabled

1: Enabled

Note: Parameter DLZ (No.1002#1) is valid when it is "0". When DLZ (No.1002#1) is "1", there is no connection with the parameter, and setting the reference position free of the link stopper is valid for all axes.

#3 HJZx Manual reference position return with reference position already set was performed.

0: With deceleration dogs

1: Using rapid traverse without deceleration dogs.

	#7	#6	#5	#4	#3	#2	#1	#0
1006			**ZMIx**		**DIAx**		**ROSx**	**ROTx**

『Way of Validating』: After power-on

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#0, #1 ROTx, ROSx Set linear or rotation axis

ROSx	ROTx	Content
0	0	Linear axis Metric/inch conversion All coordinate values are of the linear axis type. The stored pitch error compensation is of the linear axis type.
0	1	Rotary axis (type A) No metric/inch conversion The machine coordinate value displays in 0～360° cycle. The stored pitch error compensation is of the rotary axis type. Automatically return to the reference position at the direction of the reference position return (G28 and G30), the traverse amount can not exceed one turn.
1	0	Invalid setting
1	1	Rotary axis (type B) No metric/inch conversion The machine coordinate value, the relative coordinate value and the absolute coordinate value are in the linear axis, which can't display in cycle of 0～360°. The stored pitch error compensation is of the linear axis type. The cycle function and the indexing function of the rotation axis can not be used at the same time.

#3 DIAx Either a diameter or radius is set to be used for specifying the amount of travel on each axis

0: Radius

1: Diameter

#5 ZMIx The direction of reference postion return

0: Positive

1: Negative

	#7	#6	#5	#4	#3	#2	#1	#0
1008						**RRLx**	**RABx**	**ROAx**

『Way of Validating』: After power-on

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#0 ROAx The roll-over function of a rotation axis is.

0: Invalid

1: Valid

Note: ROAx is just valid for the rotary axis and parameter ROTx (No.1006#0) must be 1.

#1 RABx In absolute command, the axis rotats in the direction.

0: In which the distance to the target is shorter

1: Specified by the sign of command value

Note: RABx is valid only when parameter ROAx is 1.

#2 RRLx Relative coordinates are

0: Not rounded by the amount of the shift per one rotation

1: Rounded by the amount of the shift per one rotation

Note 1: RRLx is valid only when ROAx is 1.
Note 2: The movement amount of each turn is set by parameter No.1260.

1010	Number of CNC controlled axes(CCA)

『Way of Validating』: After power-on

『Value Range』：0～total number

Set the maximum number of axes controled by the CNC(0~ total),others are controlled by PLC.

	#7	#6	#5	#4	#3	#2	#1	#0
1015	DWT	WIC						

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#6 WIC Direct input measured values for workpiece origin offsets is

0: Enable only in a selected workpiece coordinate system

1:Enable In all coordinate systems

#7 DWT The unit of data followed P(specifying dwell time) is

0: IS-B is 1ms, IS-C is 0.1ms.

1: 1 ms

1020	Program axis name(CAN)

『Parameter Type』: Word axis

『Value Range』: 88(X), 89(Y), 90(Z), 65(A), 66(B), 67(C)

Set program name for each controled axis

Note 1: The same axial name can not be set.
Note 2: The address used by the 2nd miscellaneous function can not be taken as the axial name.

1022	Setting for each axis in basic coordinate system(ASA)

『Way of Validating』: After power-on

『Parameter Type』: Word axis

『Value Range』: 0～7

To ensure the levels of the arc interpolation, the tool offset and the tool nose radius, etc.

G17: X－Y level

G18: Z－X level

G19: Y－Z level

Set each control axis to be one of three basic axes---X, Y or an axis parallel to the X,Y,Z(. Only one axis of the basic three axes can be set: X, Y and Z; the parallel axes can be set as two more axes (which is paralleled with the basic axis).

Setting value	Meaning
0	They are neither basic three axes nor the parallel axes,
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Parallel axis of X axis
6	Parallel axis of Y axis
7	Parallel axis of Z axis

1023	**Number of servo axis for each axes (NSA)**

『Way of Validating』: After power-on

『Value Range』: 1～quantity of controlled axes

『Parameter Type』: Word axis

Set each control axis as the corresponding N^{th} servo axis. Generally, the setting value of the control axial number and that of the servo axial number are same. The so-called control axis number is to set parameter in the axis or the serial number of the signal in the axis. When the spindle is taken as the control axis, it is set as 5.

11.4 Parameters Related to Coordinate System

	#7	**#6**	**#5**	**#4**	**#3**	**#2**	**#1**	**#0**
1201	WZR					ZCL		

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#2 ZCL Local coordinate system when the manual reference return was performed is

0: Not cancel

1: Cancel

#7 WZR Upon power on the workpiece coordinate system memorized is

0: Not return to that specified by G54

1: Returned to that specified by G54

	#7	**#6**	**#5**	**#4**	**#3**	**#2**	**#1**	**#0**
1202					**RLC**	**G50**	**EWS**	**EWD**

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#0 EWD The shift direction of the workpiece coordinate system is

0: The direction specified by the external workpiece zero point offset value

1:In the opposite direction to that specified by the external workpiece zero point offset value

#1 EWS Shift value of the workpiece coordinate system and external workpiece zero point offset value are

0: stored in the separate memory area

1: stored in the same memory area (the work piece coordinate system movement amount is same as the external work piece zero point offset amount

#2 G50 If G50 command for setting a coordinate system is specified

0:G50 is executed and no alarm is issued

1:G50 is not executed and an alarm is issued

#3 RLC Local coordinate system

0:is not cancelled by reset

1:is cancelled by reset

1220	Extern workpiece zero point offset value(EWO)

『Modification authority』: Equipment management authority

『Value Range』: -9999 9999～9999 9999

『Parameter Type』: Word axis

This is one parameter to set the origin location of the work piece coordinate system (G54~G59). The parameter is the valid common offset amount for all work piece coordinate system.

Setting unit	IS-B	IS-C	Unit
Linear axis (Metric input)	0.001	0.0001	mm
Linear axis (Inch input)	0.0001	0.00001	inch
Rotary axis	0.001	0.0001	deg

1221	Workpiece zero point offset value in G54 workpiece coordinate system(WO1)

1222	Workpiece zero point offset value in G55 workpiece coordinate system(WO2)

1223	Workpiece zero point offset value in G56 workpiece coordinate system(WO3)

1224	Workpiece zero point offset value in G57 workpiece coordinate system(WO4)

1225	Workpiece zero point offset value in G58 workpiece coordinate system(WO5)

1226	Workpiece zero point offset value in G59 workpiece coordinate system(WO6)

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』: -99 999 999～+99 999 999

This is one parameter to set the origin location of the work piece coordinate system (G54~G59). The parameter is the valid common offset amount for all the work piece coordinate system.

SETTING UNIT	IS-B	IS-C	UNIT
Linear axis (Metric input)	0.001	0.0001	mm
Linear axis (Inch input)	0.0001	0.00001	inch
Rotary axis	0.001	0.0001	deg

1240	**Coordinate value of 1st Reference Position on each axis in the machine coordinate system(RF1)**
1241	**Coordinate value of 2nd Reference Position on each axis in the machine coordinate system(RF2)**
1242	**Coordinate value of 3rd Reference Position on each axis in the machine coordinate system(RF3)**
1243	**Coordinate value of 4th Reference Position on each axis in the machine coordinate system(RF4)**

『Modification authority』: Equipment management authority

『Way of Validating』: 1240 valid after power on; 1241～1243 valid immediately.

『Parameter Type』: Word axis

『Value Range』: -99 999 999～+99 999 999

Set the coordinate values from the 1st to the 4th reference positions in the mechanical coordinate system.

SETTING UNITS	IS-B	IS-C	UNITS
Machine in metric system	0.001	0.0001	mm
Inch machine	0.0001	0.00001	inch
Rotary axis	0.001	0.0001	deg

1260	**Amount of a shift per one rotation axis(PRA)**

『Modification authority』: Equipment management authority

『Way of Validating』: After power-on

『Parameter Type』: Word axis

『Value Range』: 1000～9 999 999

Set the movement amount of each turn in rotary axis.

11.5 Parameters Related to the Stroke Detection

Setting unit of stroke parameter Nos.1320~1327 is shown in the following table:

Setting unit	IS-B	IS-C	Unit
Metric machine	0.001	0.0001	mm
Inch machine	0.0001	0.00001	inch

Rotary axis	0.001	0.0001	deg

	#7	#6	#5	#4	#3	#2	#1	#0
1300	BFA	LZR	RL3			LMS		OUT

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#0 OUT Either the inside or outside of the stored stroke check 2 is set as an inhibition area specified by NO.1322, NO.1323

0:Inside

1:Outside

#2 LMS The EXLM signal for switching stored stroke check

0:Disabled

1:Enabled

Note:
Stroke detection 1 in memory type possesses the parameter of the restricted area set by two groups, signals are switched through the stroke limit in memory type and the set restricted area is selected.
(1)Restricted area I: Parameter No.1320 or No.1321
(2)Restricted area II: Parameter No.1326 or No.1327

#5 RL3 Stored stroke check 3 release signal RLSOT3 is

0: Disabled

1: Enabled

#6 LZR After power on before manual reference position return whether detect the stroke 1 in the memory type

0: Detect

1: Not detect

Note:
There isn't any connection with the setting when the absolute position encoder is being using, the power is on and the reference position is set. After power on, the stroke is directly detected in memory type.

#7 BFA Checking of stored stroke check 1 during the time from power-on to the manual reference postion return

0:The stored stroke 1 is checked

1:The stored stroke 1 is not checked

Note:
The tool stops before or after the maximum distance F/7500(mm) far away from the boundary. (F: Feedrate during reaching the boundary (Unit: mm/min)).

	#7	#6	#5	#4	#3	#2	#1	#0
1310							OT3x	OT2x

『Modification authority』: Equipment management

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#0 OT2X Whether stored stroke check 2 is checked for each axis

0: Stored stroke check 2 is not checked

1: Stored stroke check 2 is checked

#1 OT3X **Whether stored stroke check 3 is checked for each axis**

0: Stored stroke check 3 is not checked

1: Stored stroke check 3 is checked

1320	**Coordinate value I of stored stroke check 1 in the positive direction on each axis(PC1)**
1321	**NC1\tCoordinate value I of stored stroke check 1 in the negative direction on each axis(NC1)**

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Default Setting』: No.1320 is 99 999 999, No.1321 is －99 999 999

『Value Range』:－99 999 999～99 999 999

Respectively set the coordinate values of boundaries in positive and negative directions in the mechanical coordinate system in each axis stroke detection 1 in memory type. Set the outside of boundary as the restricted area to tools.

Note:

1. The axes specified by diameter are set by diameter value.

2. When (parameter No.1320) < (parameter No.1321) and the limit is infinite, it can not detect the stroke 1 in memory type. (The stroke limit switching signal in memory type is invalid.) If the absolute command is specified, the coordinate value may overflow, the normal movement can not be executed.

3. If parameter LMS (No. 1300#2) is "1", and the stroke limit switching signal in memory type EXLM is also "1", the restricted area is invalid set by the parameter. Parameter No.1326 and No.1327 set the restricted area.

1322	**Coordinate value of stored stroke check 2 in the positive direction on each axis(PC2)**
1323	**Coordinate value of stored stroke check 2 in the negative direction on each axis(NC2)**

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Default Setting』: NO.1322 is 99 999 999, NO.1323 is－99 999 999

『Value Range』: －99 999 999～99 999 999

Respectively set the coordinate values of boundaries in positive and negative directions in the mechanical coordinate system in each axis stroke detection 2 in memory type. The outside or inside of boundary is the restricted area, which is set by parameter OUT (No.1300#0).

Note:The axis specified by diameter must be set by the diameter value.

1324	**Coordinate value of stored stroke check 3 in the positive direction on each axis(PC3)**
1325	**Coordinate value of stored stroke check 3 in the negative direction on each axis(NC3)**

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Default Setting』: No.1324 is 99 999 999, No.1325 is －99 999 999

『Value Range』: －99 999 999～99 999 999

Respectively set the coordinate values of boundaries in positive and negative directions in the mechanical coordinate system in each axis stroke detection 3 in memory type. Set inside of the boundary as the restricted area to tools.

Note: The axis specified by the diameter must be set by the diameter value.

1326	**Coordinate value Ⅱ of stored stroke check 1 in the positive direction on each axis(PC12)**
1327	**Coordinate value Ⅱ of stored stroke check 1 in the negative direction on each axis(NC12)**

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Default Setting』: NO.1326 is 99 999 999, NO.1327 is －99 999 999.

『Value Range』: －99 999 999～99 999 999

Respectively set the positive and negative boundary coordinate values for each axis stroke detection 1 in memory type in the machine coordinate system. Set outside of the boundary as the restricted area. When parameter LMS (No.1300#2)is “1”, and the stroke limit switching signal EXLM (G7.6) in memory type is “1”, the restricted area is valid, but it is invalid if it is set by No.1320 and 1321.

Note:
1. The axes programmed by the diameter must be set by the diameter value.
2. The parameter is invalid when parameter LMS (No.1320#2) is “0”, or the stroke limit switching signal EXLM (G7.6) in the memory type is “0”. Then, the restricted area set by parameter No.1320 or No. 1321 is valid.

11.6 Parameters Related to Feedrate

	#7	#6	#5	#4	#3	#2	#1	#0
1401		RDR	TDR	RF0				RPD

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#0 RPD Manual rapid traverse during the period from power-on to completion of

reference position return

0: Disabled(JOG feed is performed)

1: Enabled

#4 RF0 When cutting feed rate override is 0% during rapid traverse

0:The machine tool doesn't stop moving

1:The machine tool stop moving

#5 TDR Dry run during threading or tapping:

0: Enabled

1: Disabled

#6 RDR Dry run for rapid traverse:

0: Enabled

1: Disabled

	#7	#6	#5	#4	#3	#2	#1	#0
1402						JOV		

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#2 JOV JOG override

0: Enabled

1: Disabled (fixed as 100%)

	#7	#6	#5	#4	#3	#2	#1	#0
1403	RTV							MIF

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#0 MIF Cutting feedrates at speed per minute is specified by F commands in unit of

0:1mm/min or 0.01inch/min

1: 0.001mm/min or 0.00001inch/min

#7 RTV During thread cutting cycle, the override of the tool run-out is

0: Enabled

1: Disabled

	#7	#6	#5	#4	#3	#2	#1	#0
1404						F8A	DLF	

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#1 DLF After a reference positon is set, manual reference position return performed at

0:Rapid traverse rate

1:Manual rapid traverse rate\

1: Move to the reference position

#2 F8A Valid data range for an F command in feed per minute mode

0:Ranged specified with para. MIF(No.1403#0)

1:Range refering to User Manual

SETTING UNITS	UNIT	IS-B	IS-C
Metric input	mm/min	0.001～60000	0.001～24000

Inch input	inch/min	0.00001～2400	0.00001～960
Rotary axis	deg/min	1～60000	1～24000

1410 **Dry run rate(DRR)**

『Parameter Type』: Word type

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE IS-B IS-C	DEFAULT SETTING
Metric machine	1mm/min	6～15000	1000
Inch machine	0.1inch/min		

Set the speed during dry run.

1411 **Cutting feedrate(IFV) in auto mode after power-on(initial value)（IFV）**

『Parameter Type』: Word type

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE	DEFAULT SETTING
Metric machine	1 mm/min	6～32767	1000
Inch machine	0.1 inch/min		

It doesn't require changing the cutting speed in the machine during the processing. And the cutting feedrate can be set by the parameter, then the cutting feedrate is not required to be set in the program. But the actual feedrate is limited by parameter NO.1422 which set the maximum cutting feedrate for all axes.

1420 **Rapid traverse rate (RTT)**

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE IS-B IS-C	DEFAULT SETTING
Metric machine	1 mm/min	6～60000	8000
Inch machine	0.1 inch/min		
Rotary axis	1 deg/min		

Set the rapid movement speed for each axis when the rapid movement override is 100%.

1421 **F0 rate of rapid traverse override for each axis(FOR)**

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	

Metric machine	1 mm/min	30～15000	30～12000	400
Inch machine	0.1 inch/min	30～6000	30～4800	
Rotary axis	1 deg/min	30～15000	30～12000	

Set the speed when the rapid movement override for each axis is 0.

1422 **Maximum cutting feedrate(MFR)for all axes**

『Parameter Type』: Word type

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE IS-B IS-C	DEFAULT SETTING
Metric machine	1mm/min	6～60000	8000
Inch machine	0.1inch/min		

Set the maximum cutting feedrate for all axes.

1423 **Feedrate in manual continuous feed(JFR)for each axis**

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Metric machine	1mm/min	6～32767		1000
Inch machine	0.1inch/min			
Rotary axis	1 deg/min			

Set the feedrate for each axis during continually manual feeding (JOG feeding), the actual feedrate is limited by parameter NO.1422 (the maximum cutting feedrate of all axes) .

1424 **anual rapid traverse rate(MRR)for eahc axis**

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNIT	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Metric machine	1 mm/min	0, 30～60000		8000
Inch machine	0.1 inch/min			
Rotary axis	1 deg/min			

Set rate of manual rapid traverse when the traverse override is 100%.

Set the maximum speed of MPG feeding.

Note:
If it is set as 0, use the setting value of parameter 1420.

1425 **FL rate of the reference position return for each axis(FLR)**

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Metric machine	1 mm/min	6～15000		200
Inch machine	0.1 inch/min			
Rotary axis	1 deg/min			

Set FL rate after deceleration when the reference position turn is performed for each axis.

11.7 Parameters Related to Control of Acceleration and Deceleration

	#7	#6	#5	#4	#3	#2	#1	#0
1601				**RTO**				

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#4 RTO Block overlap in rapid traverse

0: Blocks are not ovelapped in rapid traverse

1:Blocks are ovelapped in rapid traverse

	#7	#6	#5	#4	#3	#2	#1	#0
1610				**JGLx**				**CTLx**

『Modification authority』: Equipment management authority

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#0 CTLx Acceleration/deceleration in cutting feed including dry run (include feeding during dry run)

0:Exponential acceleration/deceleration is applied

1:Linear acceleration/deceleration after interpolation is applied

#4 JGLx The type of acceleration/deceleration in threading is

0: 0:Exponential acceleration/deceleration

1: 1:Linear acceleration/deceleration

1620	**Time constant T used for linear acceleration/deceleration(TT1)for each axis**

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』: 0～4000 ms

『Default Setting』: 100

Specify a time constant used for linear acceleration/deceleration in rapid traverse(0～4000ms)

1622	**Time constant for acceleration/deceleration afer interpolation in cutting feed for each axis(ATC)**

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』: 0～4000 ms

『Default Setting』: 100

Set the acceleration and deceleration for each axis cutting and feeding in exponential type, or the time constant of acceleration and deceleration in linear type after interpolation. And the detailed type is set by parameter CTLx (NO.1610#0). If CTLx sets the acceleration and deceleration in linear type after linear interpolation, the maximum time constant of acceleration and deceleration is limited in 512ms and even it exceeds 512ms, it is still dealt as 512ms.

Except the special usage of the parameter, all axes must be set as the same time constant. If the different time constants are set, the correct linear or circular can't be shaped.

1623	**FL rate of exponent acceleration/deceleration in cutting feed（FLC)for each axis**

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE IS-B / IS-C	DEFAULT SETTING
Metric machine	1 mm/min	0, 6～15000	30
Inch machine	0.1 inch/min		30
Rotary axis	1 deg/min		30

Set the low limit speed (FL speed) of acceleration and deceleration in exponential type for each axis cutting and feeding.

1624	**Time constant for acceleration/deceleration after interpolation in JOG feed(JET)for each axis**

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』: 0～4000ms

『Default Setting』: 100

Set the acceleration and deceleration in exponential type for each axis JOG feeding, and the time constant of acceleration and deceleration in linear type after interpolation.

The detailed type is set by parameter JGLx (NO.1610#4). If JGLx sets the acceleration and deceleration in linear type after interpolation, the maximum time constant of acceleration and deceleration is limited in 512ms and even it exceeds 512ms, it is dealt as 512ms.

1625	**FL rate of exponent acceleration/deceleration in jog feed（FLJ)for each axis**

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE IS-B / IS-C	DEFAULT SETTING
Metric machine	1 mm/min	0, 6～15000	30
Inch machine	0.1 inch/min		
Rotary axis	1 deg/min		

Set the low limit speed (FL speed) of acceleration and deceleration in exponential type during each

axis JOG feeding.

1626	**Time constant of acceleration/deceleration in thread cutting cycle(TET)for each axis**

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』: 0～4000ms

『Default Setting』: 100

Set the time constant of acceleration and deceleration in linear and exponential types during each axis thread cutting cycle.

1627	**FL rate of exponential acceleration/deceleration in thread cutting cycle(FLT) for each axis**

『Modification authority』: Equipment management authority

『Parameter Type』: Word axis

『Value Range』:

SETTING UNITS	VALUE UNITS	VALID RANGE		DEFAULT SETTING
		IS-B	IS-C	
Metric machine	1 mm/min	0, 6～15000	0, 6～12000	30
Inch machine	0.1 inch/min	0, 6～6000	0, 6～4800	30

Set lower limit speed (FL speed) of acceleration and deceleration in exponential type during each axis thread cutting cycle.

11.8 Parameters Related to Servo and Backlash Compensation

	#7	#6	#5	#4	#3	#2	#1	#0
1800	BDEC	BD8		RBK				

『Default Setting』: 1000 0000

#4 RBK: Backlash compensation applied separately for cutting feed and rapid traverse

0:Not performed

1:Performed

#6 BD8: Frequency of backlash compensation pulses output involved

0:Set frequency

1:1/8 of set frequency

#7 BDEC: Backlash compensation pulses

0:output in fixed pulse frequency

1:output based on acceleration/deceleration

	#7	#6	#5	#4	#3	#2	#1	#0
1811						POD		ABP

『Way of Validating』: After power-on

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#0 ABP Pulse drive mode select

0: Pulse +direction mode

1: AB phases pulse mode

#2 POD Pulse output direction select for each axis

0: Not reverse

1: Reverse

	#7	#6	#5	#4	#3	#2	#1	#0
1815			APCx	APZx				APRx

『Way of Validating』: After power-on

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#0 APRx Direction of position on absolute position detector when using absolute postion encoder

0: Not reverse

1:Reverse

#4 APZx Machine position and position on absolute position detector when the absolute position detector is used

0:Not corresponding

1:Corresponding

Note:
When use the absolute position detector, during the initial setting or after changing the absolute position encoder, the parameter must be set as 0, and connect power supply, again after power off and manually return to the reference position. Therefore, the mechanical position consists with that of the position encoder, and the parameter will be auto set as 1.

#5 APCx Position detector

0:Other than absolute position detector

1:Absolute position detector

	#7	#6	#5	#4	#3	#2	#1	#0
1816		DM3x	DM2x	DM1x				ISAx

『Way of Validating』: After power-on

『Parameter Type』: Bit axis

『Default Setting』: 0001 0001

#0 ISAx Servo ALM signal level select

0: High

1: Low

#4-#6 DM1x-DM3x: The setting of detection multiplier ratio (DMR)

SETTING VALUE			DETECTION MULTIPLIER（DMR）
DM3x	DM2x	DM1x	
0	0	0	1/2
0	0	1	1
0	1	0	3/2

0	1	1	2
1	0	0	5/2
1	0	1	3
1	1	0	7/2
1	1	1	4

1820 **Command multiplier ratio(CMR)for each axis**

『Parameter Type』: Word axis

『Value Range』:

COMMAND MULTIPLY RATIO (CMR)	VALID RANGE OF VALUE SET BY NO.1820	DEFAULT SETTING
1/2～1/27	102～127	2
1 ～ 48	2～96	

Set the command multiplier (CMR) for each axis.

1. When the command multiplier (CMR)is 1/2～1/27, the setting value = 1 / CMR＋100;

2. When the command multiplier (CMR)is 1～48, the setting value = 2×CMR.

Gear ratio output by each axis＝CMR/ DMR

Detection unit=minimum movement unit/ CMR

The relations between the setting units and the minimum movement units:

		IS－B		IS－C	
	Input	Least input increment	Least command increment	Least input increment	Least command increment
Metric machine	Metric	0.001mm（Diameter）	0.0005mm	0.0001mm（Diameter）	0.00005mm
		0.001mm（Radius）	0.001mm	0.0001mm（Radius）	0.0001mm
	Inch	0.0001 inch（Diameter）	0.0005mm	0.00001 inch（Diameter）	0.00005mm
		0.0001 inch（Radius）	0.001mm	0.00001 inch（Radius）	0.0001mm
Inch machine	Metric	0.001mm（Diameter）	0.00005 inch	0.0001mm（Diameter）	0.000005 inch
		0.001mm（Radius）	0.0001 inch	0.0001mm（Radius）	0.00001 inch
	Inch	0.0001 inch（Diameter）	0.00005 inch	0.00001 inch（Diameter）	0.000005 inch
		0.0001 inch（Radius）	0.0001 inch	0.00001 inch（Radius）	0.00001 inch
Rotary axis		0.001deg	0.001deg	0.0001deg	0.0001deg

1851 **Backlash compensation value(BCV)for each axis**

『Parameter Type』: Word axis

『Value Range』: -9999～+9999 (Detection unit)

『Default Setting』: 0

Set the backlash compensation value for each axis.

After connecting power supply, it compensates the backlash at the first time when the machine

moves in the direction opposite with that of the reference position return.

Detection units are related with parameter No.1820 (command multiplier CMR) and the minimum movement units, about the relations between the setting units and the minimum movement units, refer to parameter No.1820 introduction.

1852	Backlash compensation value used for rapid traverse(BCVR)for each axis

『Parameter Type』: Word axis

『Value Range』: -9999～+9999 (Detection units)

『Default Setting』: 0

Set the backlash compensation value during each axis rapid movement. It is valid when parameter NO.1800#4(RBK) is set as 1. It can change the backlash compensation value based on the cutting feedrate/rapid movement speed to process in higher precision.

Note:

1. Manually continuous feeding (JOG) is taken as cutting feed.

2. After connecting power supply and before the reference position return completes at the first time, it doesn't compensate the backlash in cutting feed/rapid movement. No matter the compensation value is the cutting feed or the rapid movement, it should be compensated based on parameter NO.1851.

3. When parameter NO.1800#4(RBK) is set as 1, parameter NO.1851 is the backlash compensation value of cutting feed, parameter NO.1852 is the backlash compensation value of rapid movement. When parameter NO.1800#4(RBK) is set as 0, parameter NO.1851 is the backlash compensation value of cutting feed/rapid movement.

	#7	#6	#5	#4	#3	#2	#1	#0
1853				CPF5	CPF4	CPF3	CPF2	CPF1

『Default Setting』: 0000 0111

CPF1～CPF5: he setting of pulse frequency for backlash compensation (in BCD code)

Setting frequency= (setting value +1) Kpps

CPF5	CPF4	CPF3	CPF2	CPF1	SETTING FREQUENCY (Kpps)
0	0	0	0	0	1
0	0	0	0	1	2
0	0	0	1	0	3
0	0	0	1	1	4
0	0	1	0	0	5
0	0	1	0	1	6
0	0	1	1	0	7
0	0	1	1	1	8
0	1	0	0	0	9
0	1	0	0	1	10
0	1	0	1	0	11
0	1	0	1	1	12
0	1	1	0	0	13
0	1	1	0	1	14
0	1	1	1	0	15
0	1	1	1	1	16
1	0	0	0	0	17
1	0	0	0	1	18

1	0	0	1	0	19
1	0	0	1	1	20
1	0	1	0	0	21
1	0	1	0	1	22
1	0	1	1	0	23
1	0	1	1	1	24
1	1	0	0	0	25
1	1	0	0	1	26
1	1	0	1	0	27
1	1	0	1	1	28
1	1	1	0	0	29
1	1	1	0	1	30
1	1	1	1	0	31
1	1	1	1	1	32

2071	Backlash acceleration effective duration(BAT)

『Parameter Type』: Word axis

『Value Range』: 0～100 ms

『Default Setting』: 40

Set backlash acceleration effective duration.

11.9 Parameters Related to Input/Output

	#7	#6	#5	#4	#3	#2	#1	#0
3003	ESP							

『Default Setting』: 1000 0000

#7 ESP ESP alarm signal (X0.5)

0:Alarm when the signal is 0

1:Alarm when the signal is 1

	#7	#6	#5	#4	#3	#2	#1	#0
3004			OTH					

『Default Setting』: 0010 0000

#5 OTH The overtravel limit signal is

0: Checked

1: Not checked

	#7	#6	#5	#4	#3	#2	#1	#0
3006								GDC

『Default Setting』: 0000 0000

#0 GDC As the deceleration signal of the reference position return

0: Use X signal

1: Use G196 (X signal is invalid)

#7	#6	#5	#4	#3	#2	#1	#0

3009				DECx					

『Parameter Type』: Bit axis

『Default Setting』: 0010 0000

#5 DECx: Deceleration signal of the reference position return

0: decelerate when the signal is 0

1: decelerate when the signal is 1

3010	Time lag in strobe signal MF,TF,SF (MFT)

『Value Range』: 16 ms～32767 ms

『Default Setting』: 16

Set the time required to send strobe signal MF, SF, TF, BF after the M, S, T, B codes are sent(16～32767ms)

3011	MAW\tAcceptable width(MAW) of M, T, S function completion signa(FIN)

『Parameter Type』:Word type

『Default Setting』: 16

Set the minimum signal width of the valid M, T, S and B function completion signal(FIN)(16～32767ms)

Note:
Time is set by 8ms, if the setting value is not the multiple of 8, it should be carried into the multiple of 8.

3017	Output time of reset signal(RST)

『Value Range』: 0～255

『Default Setting』: 32

Set the dwell time when the resetting signal RST is output.

RST signal output time =resetting time + the parameter value X 16ms.

3030	Allowable number of digits for M code(MCB)

『Value Range』: 2～8

『Default Setting』: 2

Set the allowable number of digits for M code(2～8)

3031	Allowable number of digits for S code(SCB)

『Value Range』: 1～5

『Default Setting』: 4

Set the allowable number of digits for S code(1～5)

Maximum 5 digits in S code are allowed.

3032	Allowable number of digits for T code(TCB)

『Value Range』: 2～8

『Default Setting』: 4

Set the allowable number of digits for T code(2～8).

11.10 Parameters Related to Display and Editing

	#7	#6	#5	#4	#3	#2	#1	#0
3101				BGD				

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#4 BGD In background editing, a program currently selected in the frontground

0: Can't be selected

1: Can be selected

	#7	#6	#5	#4	#3	#2	#1	#0
3102					CHI			

『Way of Validating』: After power-on

『Default Setting』: 0000 1000

#3 CHI Select display language

0: English

1: Chinese

Set the selected language to display.

	#7	#6	#5	#4	#3	#2	#1	#0
3104	DAC	DAL	DRC	DRL				MCN

『Default Setting』: 1100 0000

#0 MCN Machine position 0:Displayed according to the unit of output

0: Displayed according to the unit of input

(There isn't any connection with the metric system or the inch system, the metric machine displays as the metric units, the inch machine displays as the inch units.)

1: Displayed according to the unit of input

(When it is input in the metric system, display in the metric system; when it is input in the inch system, display in the inch system)

#4 DRL Relative position

0: The actual position displayed takes into acount tool offset

1: The programed position displayed does not take into acount tool offset

Note: In T serial, the movement coordinate system compensates the tool appearance, (parameter LGT (NO.5002#4) is 0), display the programming position which ignores the tool compensation (the parameter is set as 1). However, the programming position without the tool appearance compensation value can not display.

#5 DRC Relative position

0:The actual position displayed takes into acount tool nose radius compensation

1:The programed position displayed does not take into acount tool nose radius compensation

#6 DAL Absolute position

0:The actual position displayed takes into acount tool offset

1:The programed position displayed does not take into acount tool offset

Note: In T serial, the movement coordinate system compensates the tool appearance (parameter LGT (NO.5002#4) is 0), and display the programming position which ignores

the tool compensation (the parameter is set as 1). However, the programming position without the tool appearance compensation value can not display.

DAC: Absolute position

0:The actual position displayed takes into acount tool nose radius compensation

1:The programed position displayed does not take into acount tool nose radius compensation

	#7	#6	#5	#4	#3	#2	#1	#0
3107				SOR	REV	DNC		

『Modification authority』: Equipment management authority

『Default Setting』: 0001 0000

#2 DNC Upon reset, the program displayed for DNC operation is

0:Not cleared

1:Cleared

#3 REV The actual speed in feed per revolution mode is displayed in

0: mm/min or inch/min

1: mm/rev or inch/rev

#4 SOR Display of the program directory

0:Programs are listed in the order of registeration

1:Programs are listed in the order of program number

	#7	#6	#5	#4	#3	#2	#1	#0
3110						AHC		

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#2 AHC With a soft key, the alarm history

0:Can be cleared

1:Can't be cleared

	#7	#6	#5	#4	#3	#2	#1	#0
3111	NPA							

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#7 AHC Action taken when an alarm is generated or when an operator message is entered:

0: The display shifts to the alarm message screen

1: The display doesn't shift to the alarm message screen

	#7	#6	#5	#4	#3	#2	#1	#0
3114								IPC

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#0 IPC When the function key is pressed whose screen is being displayed

0:The screen is changed

1:The screen is not changed

	#7	#6	#5	#4	#3	#2	#1	#0
3202			CPD					

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#5 CPD When a NC program is deleted, a confirmation message and soft key are

0:Not output

1:Output

	#7	#6	#5	#4	#3	#2	#1	#0
3203	MCL	MER						

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#6 MER When the last block of a program has been executed in the MDI mode, the executed block is

0:Not deleted

1:Deleted

Note: Even MER is 0, when "%" (end code) is read in and executed, the program is also deleted ("%"is auto inserted at the end of the program).

#7 MCL Whether a prepared program in MDI mode is cleared by reset

0: Not deleted

1: Deleted

	#7	#6	#5	#4	#3	#2	#1	#0
3209								MPD

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#0 MPD When a subprogram is executed, the main program number is

0: Not displayed

1: Displayed

3216	**Increment in sequence numbers inserted automatically(INC)**

『Modification authority』:Equipment management authority

『Value Range』: 0～9999

『Default Setting』: 10

When the serial number (parameter SEQ(NO.0000#5) is 1) is auto inserted, it is the increment value of the serial number in each block.

11.11 Parameters Related to Programming

	#7	#6	#5	#4	#3	#2	#1	#0
3401						NCK		DPI

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0001

#0 DPI When a decimal point is omitted in an address that can include a decimal point

0:The least input increment is assumed

1:The unit of mm, inch, second is assumed

#2 NCK The same sequence number is specified twice or more in a program

0:An alarm is issued

1:Not alarm

	#7	#6	#5	#4	#3	#2	#1	#0
3402	G23	CLR		FPM				G01

『Modification authority』: Equipment management authority

『Default Setting』: 0001 0000

#0 G01 Mode entered when the power is turned on or when the control is cleared

0: G00 mode

1: G01 mode

#4 FPM When the power is turned on

0: Feed per revolution mode

1: Feed per minute mode

#6 CLR Reset button on the MDI panel, external reset signal, and emergency stop signal

0:Causes reset state

1:Causes clear state

#7 G23 **When the power is turned on**

0: G22 mode(stored stroke check on)

1: G23 mode(stored stroke check off)

	#7	#6	#5	#4	#3	#2	#1	#0
3403		AD2	CIR	RER				

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#4 RER When arc radius(R) not out-of-tolerance is so small that end point is not on the arc in arc interpolation

0: Calculate new radius for semicircle

1: Alarm is issued

#5 CIR When neither the distance(I,J,K)from the start point to the center nor an arc radius (R) is specified in circular interplation

0: The tool moves to end point by linear interplation

1: Alarm is issued

#6 AD2 Specification of the same address two or more times in a block

0: Next specification is enabled

1: Alarm

Note:
It alarms when the parameter is 1and two or two more G codes of one group are commanded in one block.

	#7	#6	#5	#4	#3	#2	#1	#0
3404	M3B	EOR	M02	M30				

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#4 M30 When M30 is executed in automatic operation

0: Control returns to the head of the program.

1: Control does not return to the head of the program.

#5 M02 When M02 is executed in automatic operation

0:Control returns to the head of the program.

1:Control does not return to the head of the program.

#6 EOR When an end-of-record mark(%) is read during program execution

0: Alarm occurs

1: No alarm occurs

#7 M3B The number of M codes that can be specified in one block

0: 1

1: Up to 3

3410	Tolerance of arc radius(CRE)

『Modification authority』: Equipment management authority

『Value Range』: 0～9999 9999

『Default Setting』: 0

Setting unit	IS－B	IS－C	Unit
Input in mm	0.001	0.0001	mm
Inch input	0.0001	0.00001	inch

Set the allowable error value of arc interpolation (G02, G03) starting point radius and its finishing point radius. P/S alarms when arc interpolation radius error is more than the limit value.

Note:
When the setting value is 0, it doesn't require checking the arc radius error.

11.12 Parameters Related to Screw Pitch Error Compensation

3620	Number of the pitch error compensation position for reference position(NPR)for each axis

『Way of Validating』: After power-on

『Parameter Type』: Word axis

『Value Range』: 0～1023

『Default Setting』: 0

3621	Number of pitch error compensation position at extremely negative position (NEN)for each axis

『Way of Validating』: After power-on

『Parameter Type』: Word axis

『Value Range』: 0～1023

『Default Setting』: 0

The parameter sets the number of the furthest screw pitch error compensation point for each axis in negative direction.

3622	Number of pitch error compensation position at extremely positive position (NEP)

『Way of Validating』: After power-on

『Parameter Type』: Word axis

『Value Range』: 0～1023

『Default Setting』: 0

The parameter sets the number of the furthest screw pitch error compensation point for each axis in positive direction.

The parameter setting value should be greater than that of parameter NO.3620.

3623	Magnification for pitch error compensation (PCM)for each axis

『Way of Validating』: After power-on

『Parameter Type』: Word axis

『Value Range』: 0～100

『Default Setting』: 0

Set the override for each axis screw pitch error compensation.

If the override is set as 1, the detection unit is same as that of compensation.

If the override is set as 0, the override is same as one when it is set as 1.

3624	Interval between pitch error compensation positions (PCI)for each axis

『Way of Validating』: After power-on

『Parameter Type』: Word axis

『Value Range』: 0～100

『Default Setting』: 0～99 999 999

『Default Setting』: 0

Setting unit	IS－B	IS－C	Unit
Metric input	0.001	0.0001	mm
Inch input	0.0001	0.00001	inch
Rotary axis	0.001	0.0001	deg

The screw pitch compensation points are distributed in equal interval, and the interval value for each axis is set respectively. The minimum value of the interval is limited and set by the following formula: the minimum value = the maximum feedrate (rapid feedrate) / 7500.

Unit: Screw pitch compensation minimum interval: mm, inch and deg.

Maximum feedrate: mm/min, inch/min and deg/min.

For example: When the maximum feedrate is 15000mm/min, the minimum value of the screw pitch error compensation interval is 2mm.

But, according to the setting override, when the absolute value of the compensation point value exceeds 100, the interval of the compensation point is magnified by the override which is calculated by the following formula.

Override = Max compensation amount (absolute value)/128 (round up the digits after the decimal point)

Screw pitch compensation minimum interval = Value, which is obtained from the above maximum feedrate X override.

Note:

The unit of the screw pitch compensation value is same as that of the detection.

The detection unit is relative with parameter No.1820 (command magnify ratio CMR) and the

minimum movement unit, about the relation between the setting units and the minimum movement units, refer to the introduction of parameter No.1820.

	#7	#6	#5	#4	#3	#2	#1	#0
3628				NPF5	NPF4	NPF3	NPF2	NPF1

『Default Setting』: 0000 0111

#0~#4 NPF1～NPF5 The setting of pulse frequency for pitch error compensation (in BCD code).

Setting frequency= (setting value +1) Kpps

NPF5	NPF4	NPF3	NPF2	NPF1	Setting frequency (Kpps)
0	0	0	0	0	1
0	0	0	0	1	2
0	0	0	1	0	3
0	0	0	1	1	4
0	0	1	0	0	5
0	0	1	0	1	6
0	0	1	1	0	7
0	0	1	1	1	8
0	1	0	0	0	9
0	1	0	0	1	10
0	1	0	1	0	11
0	1	0	1	1	12
0	1	1	0	0	13
0	1	1	0	1	14
0	1	1	1	0	15
0	1	1	1	1	16
1	0	0	0	0	17
1	0	0	0	1	18
1	0	0	1	0	19
1	0	0	1	1	20
1	0	1	0	0	21
1	0	1	0	1	22
1	0	1	1	0	23
1	0	1	1	1	24
1	1	0	0	0	25
1	1	0	0	1	26
1	1	0	1	0	27
1	1	0	1	1	28
1	1	1	0	0	29
1	1	1	0	1	30
1	1	1	1	0	31
1	1	1	1	1	32

11.13 Parameters Related to the Spindle Control

	#7	#6	#5	#4	#3	#2	#1	#0
3700							NRF	

『Modification authority』: Equipment

『Default Setting』: 0000 0000

#1 NRF The first move command(G00) after the spindle is switched to Cs axis performs

0:Positioning after returning to the reference postion

1:Normal positioning

	#7	#6	#5	#4	#3	#2	#1	#0
3705				EVS				

#4 EVS When the spindle control function is used, S codes and SF are (spindle analog output or spindle serial output)

0: Not output for an S command

1: Output for an S command

	#7	#6	#5	#4	#3	#2	#1	#0
3706							PG2	PG1

『Default Setting』: 0000 0000

#0, #1 PG2 and PG1 Gear ratio between the spindle and the position encoder.

Gear ratio=spindle speed/position encoder speed

Gear ratio	PG2	PG1
×1	0	0
×2	0	1
×4	1	0
×8	1	1

	#7	#6	#5	#4	#3	#2	#1	#0
3707							P22	P21

『Default Setting』: 0000 0000

#0, #1 P22 and P21 Gear ratio between the spindle and the second position encoder.

Gear ratio= spindle speed/position encoder speed

Gear ratio	P22	P21
×1	0	0
×2	0	1
×4	1	0
×8	1	1

Note: The parameter is valid only when multi-spindle control.

	#7	#6	#5	#4	#3	#2	#1	#0
3708		TSO					SAT	SAR

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#0 SAR The spindle speed arrival signal is

0: Not checked

1: Checked

#1 SAT Check of the spindle speed arrival signal at the start of executing the thread cutting block

0: The signal is checked only when SAR is set

1: The signal is always checked irrespective of whether SAR is set

Note: When the thread cutting block is continuously executed, the spindle speed reaching signal isn't checked in the thread cutting block after the 2nd block.

#6 TSO During a threading or tapping cycle, the spindle override is

0: Disabled(tied to 100%)

1: Enabled

Note:
In rigid tapping, the override is fixed as 100%, and there isn't any connection with the setting of the parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
3709						MSI		SAM

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#0 SAM The sampling frequency to obtain the average spindle speed

0: 4(Normally, set to 0)

1: 1

#2 MSI In multi-spindle control,the SIND signal is valid

0: Only when the first spindle is valid

1: For each spindle irrespective of whether the spindle is selected

3730	Data used for adjusting the gain of analog output of spindle speed (AGS)

『Value Range』: 700～1250

『Default Setting』: 1000

『Value unit』: 0.1%

Set data used for adjusting the gain of analog output of spindle speed. (Adjusting method)

（1） Set the standard setting value 1000,

（2） Command the spindle speed when the spindle speed analog output maximum voltage is 10V.

（3） Measure the output voltage.

（4） Set the value in the following formula in parameter No.3730:

$$\text{setting value} = \frac{10(\text{V})}{\text{measured voltage(V)}} \times 1000$$

(5) After setting the parameter, command the spindle speed analog output as the spindle speed of the maximum voltage, again, and confirm the output voltage as 10V.

3731	**Compensation value for offset voltage of analog output of the spindle speed(CSS)**

『Value Range』: -1024～+1024

Set compensation value for offset voltage of analog output of the spindle speed(-1024～1024).

1. Set the standard setting value as 0.
2. Command the analog output voltage as 0V, which is the theoretical spindle speed.
3. Measure the output voltage.
4. Set the value in the following formula in parameter No.3731.

$$\text{setting value} = \frac{-8191 \times \text{offset voltage(V)}}{12.5}$$

5. After setting the parameter, command the analog output voltage as 0V, again, which is the theoretical spindle speed and confirm the voltage as 0V.

3740	**Time elapsed prior to checking the spindle speed arrival singal(SAD)**

『Value Range』: 0～255ms

『Default Setting』: 6000

Set the time elapsed from the execution of the S function up to the checking of the spindle speed arrival signal.

3741	**Maximum spindle speed for gear 1(MSG1)**

3742	**Maximum spindle speed of gear 2 (MSG2)**

3743	**Maximum spindle speed of gear 3 (MSG3)**

3744	**Maximum spindle speed of gear 4 (MSG4)**

『Default Setting』: 6000

『Value Range』: 0～32767r/min

The parameter sets the maximum spindle speed of each gear.

3770	**Axis as the calculation reference in constant surface speed control(ACS)**

『Default Setting』: 0

『Value Range』: 0, 1～quantity of the control axes

Set the axis as the calculation reference in constant surface speed control

Note:
When it is set as 0, default X axis. Then, P value commanded in G96 block is not significant to the constant surface speed.

3771	**Minimum spindle speed in constant surface speed control(G96) (CFL)**

『Value Range』: 0～32767r/min

『Default Setting』: 0

The parameter sets the minimum spindle speed in the constant surface speed control. During the constant surface speed control (G96) , if the spindle speed is lower than the speed set by the parameter, it is limited in the parameter speed.

3772	Maximum spindle speed(MSS)

『Value Range』: 0～32767r/min

『Default Setting』: 6000

The parameter sets the maximum spindle speed. The actual spindle speed is limited by the maximum speed set by the parameter when the commanded spindle speed exceeds the maximum spindle speed, or the spindle speed after override exceeds the maximum spindle speed.

Note:

1. When the constant surface speed controls, no matter whether G96 or G97 is commanded, the spindle speed is limited by the maximum spindle speed.

2. When the setting value is 0, it is not limited by the speed.

3. When PLC controls the spindle speed, the parameter is invalid and the spindle speed isn't limited by the maximum speed.

4. When multi-spindle control, the maximum speed of each spindle is set through the following parameters:

The maximum speed of the 1st spindle is set by parameter NO.3772.

The maximum speed of the 2nd spindle is set by parameter NO.3802.

3773	Quantity of the spindle encoder pulses (CNT)

『Way of Validating』: After power-on

『Value Range』: 100～9999

『Default Setting』: 1024

The parameter sets the quantity of the spindle encoder pulses.

3802	Maximum speed of the 2nd spindle (MSS2)

『Value Range』: 0～32767r/min

『Default Setting』: 6000

The parameter sets the maximum speed of the 2nd spindle. The actual spindle speed is limited by the maximum speed set by the parameter when the commanded spindle speed exceeds the maximum spindle speed, or the spindle speed after override exceeds the maximum spindle speed.

Note:

1. When the multi-spindle controls, the parameter is valid.

2. When the constant surface speed controls, no matter whether G96 or G97 is commanded, the spindle speed is limited by the maximum speed.

3. When the setting value is 0, parameter NO.3772 is valid (the maximum speed of the 1st spindle). When parameter NO.3772 is 0, the spindle speed is not limited.

4. When PLC controls the spindle speed, the parameter is invalid and the spindle speed isn't limited by the maximum speed.

3803	Quantity of the 2nd spindle encoder pulses (CNT2)

『Way of Validating』: After power-on

『Default Setting』: 1024

『Value range』: 100～9999

The parameter sets the quantity of the 2nd spindle encoder pulses.

3811	Spindle maximum speed of the 2nd spindle gear 1 (M2G1)

3812	Spindle maximum speed of the 2nd spindle gear 2 (M2G2)

『Default Setting』: 6000

『Value Range』: 0～32767r/min

The parameter sets the maximum speed of each gear in the 2nd spindle.

Note: It is for multi-spindle control.

3830	Gain regulation data of the 2nd spindle speed analog output（AGS2）

『Modification authority』: Machine

『Value range』: 700～1250

『Data unit』: 0.1%

『Default setting』: 1000

Set the gain regulation data of the 2nd spindle speed analog output.

Setting method:

(1) Set the standard setting value 1000.

(2) Command the spindle speed when the spindle speed analog outputs max. 10V.

(3) Measure the output voltage.

(4) Set the following value based on No.3830:

$$\text{setting value} = \frac{10(\text{V})}{\text{measured voltage(V)}} \times 1000$$

(5) After a parameter is set, the spindle speed is commanded when the spindle speed analog output is the max. voltage, and the output voltage should be 10V.

3831	Compensation value of the 2nd spindle speed analog outputting offset voltage（CSS2）

『Modification authority』: Machine

『Value range』: -1024～+1024

『Default setting』: 0

Set the compensation value of the 2nd spindle speed analog outputting offset voltage.

Setting method:

(1) Set the standard setting value 0.

(2) Command the theory spindle speed when the analog output voltage is 0V .

(3) Measure the output voltage.

(4) Set the following value based on:

$$\text{setting value} = \frac{-8191 \times \text{offset voltage(V)}}{12.5}$$

(5) After a parameter is set, the theory spindle speed is commanded when the analog output voltage is 0V. And the voltage should be 0V.

3900	Servo axis number to execute the interpolation with Cs contour controlled axis（CSA1）
3910	Servo axis number to execute the interpolation with Cs contour controlled axis（CSA2）
3920	Servo axis number to execute the interpolation with Cs contour controlled axis（CSA3）

『Modification authority』: Machine

『Value range』: 0～controllable axes

『Default setting』: 0

The above 3 parameters set the servo axis numbers to execute the interpolation with Cs contour controlled axis.

Note: It is set to 0 when there is no servo axis to execute the interpolation with Cs contour controlled axis.

11.14 Parameters Related to the Tool Compensation

	#7	#6	#5	#4	#3	#2	#1	#0
5001		EVO		EVR				

『Modification authority』: Equipment management authority

『Default setting』: 0000 0000

#4 EVR When in tool radius compensation mode, the compensation amount is changed

0: A block specifying the next T code and subsequent blocks become valid

1: A block to be buffered next and subsequent blocks become valid

#6 EVO When in tool offset compensation mode, the compensation amount is changed

0: A block specifying the next T code and subsequent blocks become valid

1: A block to be buffered next and subsequent blocks become valid

	#7	#6	#5	#4	#3	#2	#1	#0
5002		LWM		LGT		LWT		LD1

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#0 LD1 Offset number of tool offset

0：Specified using the lower two digits of a T code

1: Specified using the lower one digit of a T code

#2 LWT Tool wear compensations is performed by

0: Shifting the coordinate system 1: Moving the tool (there isn't any connection with LWM, and compensate in the block of T code)

#4 LGT Tool offset compensation

0: Compensated by the shift of the coordinate system (there isn't any connection with LWM, and compensate in the block of T code)

1: Compensated by the tool movement

#6 LWM Tool offset(when LGT=1)

0: Is done in the T code block

1: Is done together with the axis movement

Note: When LGT is 0, the offset is executed in T code block, and there isn't any connection with the parameter.

	#7	#6	#5	#4	#3	#2	#1	#0
5003		LVC				CCN		

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#2 CCN When automatic reference postion return(G28) is specified in tool nose radius compensation

0: Comoensation is cancelled in movement to the intermidiate position

1: Comoensation is not cancelled in movement to the intermidiate position,but cancelled in movement to the reference postion

#6 LVC Offset value of tool offset 0 except in MDI mode Tool offset value is

0: Not cleared but held by reset

1: Cleared by reset

	#7	#6	#5	#4	#3	#2	#1	#0
5004							ORC	

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#1 ORC Tool offset value

0: Set by the diameter specification(Can be set in only the axis under diameter programming)

1: Set by the radius specification

	#7	#6	#5	#4	#3	#2	#1	#0
5005						PRC		

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#2 PRC iDirect input of tool offset value

0: Not use a PRC signal

1: Use a PRC signal

	#7	#6	#5	#4	#3	#2	#1	#0
5006							TGC	OIM

『Modification authority』: Equipment management authority

『Way of Validating』: After power-on

『Default Setting』: 0000 0000

#0 OIM When the unit is switched between the inch and metric systems, automatic tool offset value conversation is

0: Not performed

1: Performed

#1 TGC When a T code is specified in G50, G04 or G10

0: No alarm occurs

1: Alarm occurs

	#7	#6	#5	#4	#3	#2	#1	#0
5008		CNS	CNF	MCR	CNV		CNC	CNI

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#0 CNI Interference check for tool nose radius compensation is

0: Performed

1: Not performed

#1 CNC: During interference check of tool nose radius compensation, when the direction of movement after application of offset differs from the programed direction by between 90 and 270 degrees

0: An alarm is issued

1: No alarm is issued

#3 CNV Interference check and vector erasure of tool nose radius compensation are

0: Performed

1: Not performed

#4 MCR **If G41/G42(tool nose radius compensation) is specified in MDI mode, an alarm is**

0: Not raised

1: Raised

Note: In MDI mode, the tool nose radius isn't compensated even it is set by the parameter.

#5 CNF Interference check for tool nose radius compensation when machining the inner side of full circle

0: An alarm is issued

1: No alarm is issued

#6 CNS As a step is smaller than the tool radius compensation, interference check of tool nose radius compensation

0: An alarm is issed

1: No alarm is issued

5010	Limit value that ignore the vector when the tool moves on the outside of the corner during tool nose radius compensation(CLV)

『Modification authority』: Equipment management authority

『Value Range』: 0～16383

SETTING UNITS	IS-B	IS-C	UNITS
Metric input	0.001	0.0001	mm
Inch input	0.0001	0.00001	inch

『Default Setting』: 0

Set the limit value that ignores the slight move on the outside of the corner during tool nose radius compensation(0～16383)

5013	Maximum value of tool wear compensation (MTW)

『Modification authority』: Equipment management authority

『Default Setting』: 10

『Value Range』:

SETTING UNITS	IS-B	IS-C	UNITS
Metric input	0.001	0.0001	mm
Inch input	0.0001	0.00001	inch

SETTING RANGE	IS-B	IS-C
Metric input	0～9 999 999	0～99 999 999
Inch input		

The parameter sets Set the maximum allowable tool wear compensation value.

When the set absolute value of the tool wearing compensation value exceeds the maximum value, it alarms:

Input from MDI……alarm: too many digits. Exceed range (XXXX——XXXX) (input range is in the bracket).

Input through G10……alarm: The offset value input by G10 is out of the specified range.

11.15 Parameters Related to the Canned Cycle

The setting unit of canned cycle parameter is shown as follows:

	IS-B	IS-C	UNITS
Metric input	0.001	0.0001	mm
Inch input	0.0001	0.00001	inch

11.15.1 Parameter of the Drilling Canned Cycle

	#7	#6	#5	#4	#3	#2	#1	#0
5102							**MRC**	

『Modification authority』:Equipment management authority

『Default Setting』: 0000 0000

#1 MRC A target figure other than monotonically increasing or monotonically decreasing in G71 and G72 or that on Z axis in G73

0: No alarm is issued

1: An alarm is issued

	#7	#6	#5	#4	#3	#2	#1	#0
5104						**FCK**		

『Modification authority』:Equipment management authority

『Default Setting』:0000 0000

#2 FCK : The machining profile in multiple repetitive cycle(G71,G72,G73) is

0: Not checked

1: Checked

5110	**C-axis clamp M code in drilling canned cycle(CMD)**

『Modification authority』:Equipment management authority

『Default Setting』: 0

『Value Range』: 0～99

Set M code, which can lock C axis, in the canned cycle of drilling holes.

11.15.2 Parameters Related to the Thread Cutting Cycle

5130	Chamfering distance in the thread cutting cycles (G76,G92)(THD)

『Modification authority』:Equipment management authority

『Default Setting』: 0

『Value Range』: 0～99× (0.1 screw pitch)

The parameter sets the beveling value of G76 and G92 thread cutting cycle.

11.15.3 Parameters Related to the Combined Canned Cycle

5132	Depth of cut in multiple repetitive canned cycles G71,G72(THC)

『Modification authority』:Equipment management authority

『Default Setting』: 1000

『Value Range』: 0～99 999 999

Set the cutting value of G71 and G72 combined canned cycle.

	IS-B	IS-C	UNITS
Input in metric system	0.001	0.0001	mm
Input in inch system	0.0001	0.00001	inch

5133	Escape in multiple repetitive canned cycles G71,G72(MCE)

『Modification authority』:Equipment management authority

『Default Setting』: 0

『Value Range』: 0～99 999 999

Set the run-out value of G71 and G72 combined canned cycle.

5135	Escape in multiple repetitive canned cycle G73 in X-axis direction(G73XE)

5136	Escape in multiple repetitive canned cycle G73 in Z-axis direction(G73ZE)

『Modification authority』:Equipment management authority

『Default Setting』: 0

『Value Range』: -99 999 999～99 999 999

Set the run-out value of G73 combined canned cycle along with X and Z axes direction

5137	Division count in multiple repetitive canned cycle G73(G73DC)

『Modification authority』:Equipment management authority

『Default Setting』: 1

『Value Range』: 1～99 999 999

Set the partition times of G73 combined canned cycle.

5139	Return in multiple repetitive canned cycle G74,G75(G74G75R)

『Modification authority』:Equipment management authority

『Default Setting』: 0

『Value Range』: 0～99 999 999

Set the reversal value of G74 and G75 combined canned cycle.

SETTING UNITS	IS-B	IS-C	UNITS
Metric input	0.001	0.0001	mm
Inch input	0.0001	0.00001	inch

5140	Minimum depth of cut in multiple repetitive canned cycle G76(G76MID)

『Modification authority』:Equipment management authority

『Default Setting』: 0

『Value Range』: 0～99 999 999

Set the minimum depth of cut in multiple repetitive canned cycle G76.

SETTING UNITS	IS-B	IS-C	UNITS
Metric input	0.001	0.0001	mm
Inch input	0.0001	0.00001	inch

5141	Finishing allowance of G76 combined canned cycle (G76FA)

『Modification authority』:Equipment management authority

『Default Setting』: 500

『Value Range』: 1～99 999 999

Set the finishing allowance in multiple repetitive canned cycle G76.

5142	Finishing cycle times of G76 combined canned cycle (G76FC)

『Modification authority』:Equipment management authority

『Default Setting』: 1

『Value Range』: 1～99

Set the repetition count of final finishing in multiple repetitive canned cycle G76.

5143	Tool nose angle in multiple repetitive canned cycle G76(G76TNA)

『Modification authority』:Equipment management authority

『Default Setting』: 60

『Value Range』: 0～99 (deg)

Set the tool nose angle in multiple repetitive canned cycle G76.

11.16 Parameters Related to the Rigid Tapping

	#7	#6	#5	#4	#3	#2	#1	#0
5200		FHD		DOV		CRG		G84

『Modification authority』:Equipment management authority

『Default Setting』: 0000 0000

#0 G84 Method for specifying rigid tapping

0: A M code specifying the rigid tapping mode is specified

1: G84/G88 is used to specify rigid tapping mode

#2 CRG Rigid mode when a rigid mode cancel command is specified:

0:Cancelled after signal RGTAP(G61.0) is set to 0

1:Cancelled before signal RGTAP(G61.0) is set to 0

#4 DOV Override during extraction in rigid tapping

0: Invalidated

1: Validated, the override value is set by para NO.5211

#6 FHD Feed hold and single block in rigid tapping:

0: Invalidated

1: Validated

	#7	#6	#5	#4	#3	#2	#1	#0
5201	TXZ	TDK				TDR		

『Modification authority』:Equipment management authority

『Default Setting』: 0000 0000

#2 TDR: Cutting time constant in rigid tapping:

0: Is the same during cutting and extraction

1: Not the same during cutting and extraction

#6 TDK: Specify K in tapping command

0: Take it as the cycle times

1: Ignore

#7 TXZ: Non-tapping axis is taken as the orientation in tapping command

0: Allow to use

1: Alarm

5210	Rigid tapping mode specification M code(RTMC)

『Modification authority』:Equipment management authority

『Default Setting』: 29

『Value Range』: 0～255

M code is set to specify the rigid tapping method. When it is set as 0, CNC takes it as M29.

5211	Override value during rigid tapping extraction(RTOV)

『Modification authority』:Equipment management authority

『Default Setting』: 120

『Value unit』: 1% or 10%

『Value Range』: 0～200

Set the override value during rigid tapping extraction(0～200), valid only when DOV(NO.5200#4) is set to 1.

5241	Maximum spindle speed in rigid tapping(RTMS)

『Modification authority』:Equipment management authority

『Default Setting』: 1000

『Value Range』: 0～9999

Set the maximum spindle speed in rigid tapping.

5261	The linear acceleration/deceleration time constant for spindle and tapping axis(RTLT) in rigid tapping

『Modification authority』:Equipment management authority

『Default Setting』: 200

『Value Range』: 0～4000ms

During the rigid tapping, the time constant of linear acceleration or deceleration of the spindle and the tapping axis is the time (parameter NO.5241) of the spindle maximum speed when the spindle reaches the rigid tapping. The actual time is the ratio between the specified spindle speed and the maximum speed multiplies by the parameter.

5271	Time constant for spindle and tapping axis in extraction operation(RTET)

『Modification authority』:Equipment management authority

『Default Setting』: 200

『Value Range』: 0～4000ms

Set the time constant of linear acceleration or deceleration of the spindle and the tapping axis during the rigid tapping run-out. The parameter is valid only when parameter TDR (NO.5201 BIT2) is set as 1.

11.17 Parameters Related to the Polar Coordinate Interpolation

	#7	#6	#5	#4	#3	#2	#1	#0
5450							AFC	

『Modification authority』:Equipment management authority

『Default Setting』: 0000 0000

#0 AFC: In polar coordinate interpolation mode, automatic override and feedrate clamp are ?

0:Not performed

1:Performed

Note:In the polar coordinate interpolation mode, the more closely the tool is near to the work piece center, the bigger the speed vector of the rotary axis is. If the center part exceeds the maximum cutting speed (parameter NO.5462), the servo (NO.411) alarms. Auto feedrate override and auto feedrate limit function auto controls the feedrate, then, the speed vector of the rotary axis doesn't exceed the maximum cutting feedrate.

5460	Axis(linear axis) specification for polar coordinate interpolation

5461	Axis(rotary axis) specification for polar coordinate interpolation

『Value Range』: 1～quantity of the control axes

『Default Setting』: NO.5460 is 0；NO.5461 is 5

Set control axis number of rotary axis to execute polar interpolation.

5462	Maximum cutting feedrate during polar coordienate interpolatoin(MFI)

『Default Setting』: 8000

『Value Range』:

	IS-B	IS-C	UNITS
Metric machine	0, 6～24 000	0, 6～10 000	mm/min
Inch machine	0, 6～9 600	0, 6～4 800	inch/min
Rotary axis	0, 6～24 000	0, 6～10 000	deg/min

Set the valid maximum feedrate of the polar coordinate interpolation. If the commanded speed is greater than the value, the speed is limited by the maximum one. When the parameter is set as 0, the speed in the polar coordinate interpolation is limited by the maximum cutting feedrate (parameter NO.1422) value.

5463	Allowable automatic override percentage in polar coordinate interpolation(API)

『Value Range』: 1～quantity of the control axes

『Default Setting』: 0

『Value Range』: 0～100 (%)

When the polar coordinate interpolation is set, the percentages of the auto override are allowed to limit the cutting feedrate of the rotary axis.

The allowable speed of the rotary axis = Maximum cutting feedrate X override percentage

In polar coordinate interpolation, the more closely the tool is near to the work piece center, the bigger the speed vector of the rotary axis is. When it exceeds the allowable speed, the feedrate automatically multiplies by the override value calculated through the following formula:

Override = Allowable speed of the rotary axis/the speed vector of the rotary axis X 100%

If the revolving speed after timing the override still exceeds the allowable speed, the feedrate is limited in the allowable maximum cutting feedrate (auto speed limit function) .

Note: When the parameter value is set as 0, it is taken as 90%;
To limit the auto speed override and the auto speed, the parameter AFC (NO.5450#1) is set as 1.

11.18 Parameters Related to the User Macro Program

	#7	#6	#5	#4	#3	#2	#1	#0
6000			SBM					G67

『Modification authority』:Equipment management authority

『Default Setting』: 0000 0000

#0 G67 If G67 is specified while G66 is not set

0: An alarm is issued

1: The specification G67 is ignored.

#5 SBM Custom macro statement

0:Not stop the single block

1:Stops the single block

	#7	#6	#5	#4	#3	#2	#1	#0
6001	CLV	CCV						

『Modification authority』:Equipment management authority

『Default Setting』: 0100 0000

#6 CCV Custom macro's common variables Nos.100～199：

0:Cleared to vacant by reset

1:Not cleared by reset

Note: In MDI mode, the macro public variables are not cleared after reset.

#7 CLV Custom macro's local variables Nos.1～13

0: Cleared to vacant by reset

1: Not cleared by reset

	#7	#6	#5	#4	#3	#2	#1	#0
6004							MFZ	NAT

『Modification authority』:Equipment management authority

『Default Setting』: 0000 0000

#0 NAT Specification of the results of custom macro function ATAN & ASIN

0:The result is 0～360 & 270～90

1:The result is -180～180 & -90～90

#1 MFZ If the angle of a custom macro operation command SIN, COS or TAN is 1.0X(-108) or below, the result is

0:Handled as underflow

1:Normalized to 0

11.19 Parameters Related to Skip Function

	#7	#6	#5	#4	#3	#2	#1	#0
6200	SKF						SK0	

『Default Setting』: 0000 0000

SK0: Specify whether the skip signal is made valid

0:Skip signal is valid when the signal is set to 1

1:Skip signal is valid when the signal is set to 0

SKF: Dry run and override for G31 skip command

0: Disabled

1: Enabled

	#7	#6	#5	#4	#3	#2	#1	#0
6210		MDC						

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#6 MDC The value of the automatic tool compensation is

0: Added to current tool offset

1: Subtracted from current tool offset

	#7	#6	#5	#4	#3	#2	#1	#0
6240	IGA							AE0

『Way of Validating』: After power-on

『Default Setting』: 0000 0000

#0 AE0 Measurement position arrival is assumed when the automatic tool compensation signal(X3.6) and XAE2(X3.7) is

0:1

1:0

#7 IGA Automatic tool compensation is:

0:Enabled

1:Disabled

6241	**Feedrate during measurement of automatic tool compensation(used with signal XAE1)**

6242	**Feedrate during measurement of automatic tool compensation(used with signal XAE2)**

『Value setting』:

SETTIN UNIT	VALUE UNIT	VALID RANGE (IS-B/ IS-C)	DEFAULT
Metric	1mm/min	6～15000	1000
Inch	0.1inch/min		

These two parameters set the feedrate during automatic tool compensation.

Note: When the setting value of parameter No. 6242 is valid, the setting value of parameter No. 6241 is valid too.

6251	**γ value on X axis during automatic tool compensation(ATOR1)**

6252	**γ value on Z axis during automatic tool compensation(ATOR2)**

『Modification authority』: Equipment management authority

『Default Setting』: 1000

『Value range』: 1～99999999

These two parameters set the γ value in tool compensation function in sequence.

Note: The value is set in radius no matter diameter or radius programming is specified.

6254	**ε value on X axis during automatic tool compensation(ATOE1)**

6255	**ε value on Z axis during automatic tool compensation(ATOE2)**

『Modification authority』: Equipment management authority

『Value range』: 1～99999999

SETTING UNIT	IS-B	IS-C	unit
Linear axis (metric	0.001	0.0001	mm

input)			
Linear axis (inch input)	0.0001	0.00001	inch
Rotary axis	0.001	0.0001	deg

These two parameters set the ε value in tool compensation function in sequence.

Note: The value is set in radius no matter diameter or radius programming is specified

11.20 Parameters Related to Graphic Display

	#7	#6	#5	#4	#3	#2	#1	#0
6550					**DPA**			

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#3 DPA In the graphic display interface, the current position displays

0: Display the actual position including the tool compensation and offset

1: Display the programming position excluding the tool compensation and offset

11.21 Parameters Related to Run Hour and Parts Count Display

	#7	#6	#5	#4	#3	#2	#1	#0
6700							**PRT**	**PCM**

『Modification authority』: Equipment management authority

『Default Setting』: 0000 0000

#0 PCM M code that counts the total number of machined parts and the number of machined parts

0: M02, or M30, or M code specified by para No.6710

1: Only M code specified by para No.6710

#1 PRT Upon reset, signal PRTSF(F62.7), which indicates that a required number of parts has been reached

0: Turned off

1: Not turned off

6710	**M code that counts the total number of machined parts and the number of machined parts(MPC)**

『Value Range』: 0～9999

『Default Setting』: 0

The machine program executes M codes set by the parameter, total quantity of the processing parts and quantity of the processing parts plus 1, respectively.

Note:
When the setting value is 0, it is invalid (M00 can't count the parts). And it can't be set as 98 and 99, neither.

6713	**Number of required parts(RPM)**

『Value Range』: 0～9 999

『Default Setting』: 0

When the quantity of the processing parts equals to that of the parts required being processed, the signal PRTSF (F62.7) of the enough quantity of the required parts outputs to PLC. However, if the quantity is 0, it is regarded as infinitely great, not output to PRTSF.

11.22 Parameters Related to MPG Feed

	#7	#6	#5	#4	#3	#2	#1	#0
7100				**HPF**				**JHD**

『Default Setting』: 0000 0000

#0 JHD Manual handle feed in JOG feed mode and incremental feed in the manual handle feed

0: Invalid

1: Valid

	JHD=0		JHD=1	
	JOG MODE	MPG MODE	JOG MODE	MPG MODE
JOG feeding	O	×	O	×
MPG feeding	×	O	O	O
Increment feeding	×	×	×	O

#4 HPF When a Manual handle feed exceeding the rapid traverse rate is issued

0: The rate is clamped at the rapid traverse rate,and the handle pulses corresponding to the excess are ignored

1: the exceeded be not ignored

	#7	#6	#5	#4	#3	#2	#1	#0
7102								**HNGx**

『Parameter Type』: Bit axis

『Default Setting』: 0000 0000

#0 HNGx: Axis movement direction for rotation direction of manual pulse generator

0: Same in direction

1: Reverse in direction

	#7	#6	#5	#4	#3	#2	#1	#0
7103						**HNT**		

『Modification authority』: System

『Default setting』: 0000 0000

#2 HNT The manual handle feed/incremental feed magnification is

0: Multiplied by 1

1: Multiplied by 10

7110	**Number of manual pulse generators used(NMP)**

『Value Range』: 0～2

『Default Setting』: 1

Set the number of manual pulse generators.

7113	**Manual handle feed magnification M(MFM)**

『Value Range』: 1～127

『Default Setting』: 100

Set the magnification when manual handle feed movement selection signal MP1=0,MP2=1

7114	Manual handle feed magnification N(MFN)

『Value Range』: 1～1000

『Default Setting』: 1000

Set the magnification when manual handle feed movement selection signal MP1=1, MP2=1.

MOVEMENT VALUE SELECTING SIGNAL		MOVEMENT VALUE (MPG FEEDING)
MP2	MP1	
0	0	Minimum setting unit * 1
0	1	Minimum setting unit * 10
1	0	Minimum setting unit * M
1	1	Minimum setting unit * N

7117	Allowable numbers of pulses that can be accumulateed during manual handle feed(APM)

『Value Range』: 0～99999999

『Default Setting』: 10000

When MPG feeding instance exceeds the rapid movement speed, the pulse exceeding the rapid movement is not canceled but saved. The parameter sets the allowable value of the memory capacity.

Note:
When overrides, such as X100 or more than it, are selected, MPG rapidly turns round. MPG feeding is more than the rapid movement speed, the speed is limited by the rapid movement speed. The pulse exceeding the rapid movement speed is ignored, therefore, the scale value of MPG doesn't comply with the actual movement value. Then, If the allowable value is preset in the parameter, the pulse exceeding the rapid movement speed is not canceled, but saved in CNC temporarily (the part exceeding the allowable value is ignored). When MPG revolving speed becomes slower or the revolving stops, the saved pulse changes into the movement command and outputs. Pay attention to it if the allowable value is set too big, even MPG is stopped revolving, CNC won't stop until the remaining pulse is completed.

11.23 Parameters Related to PLC Axis Control

	#7	#6	#5	#4	#3	#2	#1	#0
8001			NCC		RDE	OVE		MLE

『Default Setting』: 0000 0000

#0 MLE Whether all axis machine lock signal MLK is valid for PLC-controlled axes

0: Valid

1: Invalid

#2 OVE Signals related to dry run and override used in PLC axis control

0: Same signals as those used for the CNC

1: Signal specific to the PLC

#3 RDE Whether dry run is valid for rapid traverse in PLC axis control

0: Invalid

1:Valid

#5 NCC When a travel command is issued for the PLC-controlled axis according to the program

0: An alarm is issued when PLC controls the axis with an axis control comang.When the PLC does not control the axis,a CNC command is enabled

1: An alarm is issued unconditionally.

	#7	#6	#5	#4	#3	#2	#1	#0
8002	FR2	FR1	PF2	PF1	F10		DWE	RPD

『Default Setting』: 0000 0000

#0 RPD Rapid traverse rate for PLC-controlled axes

0: Feedrate specified with para NO.1420

1: Feedrate specified with the feedrate data in an axis control command

#1 DWE Minimum time which can be specified in a dwell command in PLC axis control when the increment system is IS-C

0: 1ms

1: 0.1ms

#3 F10 Least increment for the feedrate for cutting feed(per minute) in PLC axis control

F10	Metric input	Inch input
0	1mm/min	0.01inch/min
1	10mm/min	0.1inch/min

#4,#5 PR1, PR2 In PLC axis control, the least increment unit of cutting feed

PF2	PF1	Speed
0	0	1/1
0	1	1/10
1	0	1/100
1	1	1/1000

#6,#7 FR1, FR2 the feedrate unit of feed per rotation in PLC axis control

FR2	FR1	Metric input	Inch input
0	0	0.0001mm/rev	0.000001inch/rev
1	1		
0	1	0.001mm/rev	0.00001inch/rev
1	0	0.01mm/rev	0.0001inch/rev

	#7	#6	#5	#4	#3	#2	#1	#0
8003								PIM

『Way of Validating』: After power-on

『Default Setting』: 0000 0000

#0 PIM If PLC control axis is linear axis, the control commands are

0: Affected by inch system/metric system

1: Not affected by inch system/metric system

	#7	#6	#5	#4	#3	#2	#1	#0
8004	NDI	NCI	DSL			JFM	NMT	CMV

『Default Setting』: 0000 0000

#0 CMV According to the commands sent by CNC, PLC sends the axis control command after moving along the axis and before receiving the command signal of the miscellaneous function.

0: P/S No.130 alarms

1: The axis is processed as one PLC axis and is executed the set movement.

#1 NMT: When PLC is processing one control command of some axis, and CNC sends another command to command the axis, PLC control axis is still

0: P/S No.133 alarms

1: Not alarm

#2 JFM Feedrate units of continuous feeding (06h) of PLC control axis

INCREMENT SYSTEM	JFM	METRIC INPUT	INCH INPUT	ROTARY AXIS
IS-B	0	1mm/min	0.01inch/min	1deg/min
	1	200mm/min	2.00inch/min	200deg/min
IS-C	0	0.1mm/min	0.001inch/min	0.1deg/min
	1	20mm/min	0.200inch/min	20deg/min

#5 2DSL When selecting the axes controlled by PLC is forbidden, if the axes are tried to exchange

0: Failed and P/S No.139 alarms

1: Axes, without commanding the channel, are executed exchanging

#6 NCI In axis control by the PLC, a position check at the time of deceleration is

0: Performed

1: Performed

#7 NDI: When PLC control axis selects the diameter programming, under PLC axis control

0: The radius programming specifies the movement distance and the feedrate

1: The diameter programming specifies the movement distance and the feedrate

	#7	#6	#5	#4	#3	#2	#1	#0
8005							**CDI**	

『Default Setting』: 0000 0000

#1 CDI For PLC axis control, when diameter programing is specified for a PLC-controlled axis

0: The amount of travel is specified with a radius

1: The amount of travel is specified with a diameter

8010	**Selection of the DI/DO group for each axis controlled by PLC(EPSA)**

『Parameter Type』: Word type

『Default Setting』: 0

『Value Range』: 0～4

Each DI/DO group controlled by each PLC axis, which is shown as the following list:

NUMERICAL VALUE	**REMARK**
0	The axis is not controlled by PLC

1	DI/DO in group A is used
2	DI/DO in group B is used
3	DI/DO in group C is used
4	DI/DO in group D is used

8022	**Upper-limit rate of feed per revolution during PLC axis control(EPMF)**

『Parameter Type』: Word type

『Default Setting』: 6

『Value Range』:

INCREMENT SYSTEM	VALUE UNITS	VALID VALUE RANGE	
		IS-B	IS-C
Metric machine	1mm/min	6~15000	6~12000
Inch machine	0.1inch/min	6~6000	6~4800
Rotary axis	1deg/min	6~15000	6~12000

Set the upper-limit rate of feed per revolution during PLC axis control

8028	**Linear acceleration/deceleration time constant of speed command for PLC axis control(EPAT)**

『Parameter Type』: Word axis

『Default Setting』: 200

『Value Range』: 0~3000ms

Set the time required for the servo motor rotation speed to increase or decrease in JOG feed.

Note: If it is set to "0", the system doesn't control the acceleration and deceleration.

11.24 Parameters Related to Basic Function

8130	**Total number of controlled axes(TCA)**

『Way of Validating』: After power-on

『Default Setting』：2

『Value Range』：2～5

Set the total number of controlled axes by the CNC.

	#7	#6	#5	#4	#3	#2	#1	#0
8131								HPG

『Way of Validating』: After power-on

『Default Setting』: 0000 0001

#0 HPG Manual handle feed is

0: Not used

1: Used

	#7	#6	#5	#4	#3	#2	#1	#0
8132								TLF

『Way of Validating』: After power-on

『Default Setting』: 0000 0000

#0 TLF : Tool life management is

0: Not used

1: Used

	#7	#6	#5	#4	#3	#2	#1	#0
8133					MSP	SCS	AXC	SSC

『Way of Validating』: After power-on

『Default Setting』: 0000 0001

#0 SSC Constant surface speed control is

0: Not used

1: Used

#1 AXC Spindle positioning is

0: Not used

1: Used

#2 SCS CS contour control is 0

0: Not used

1: Used

#3 MSP ulti-spindle control is 0

0: Not used

1: Used

11.25 Parameters Related to GSK-CAN Communication Function

	#7	#6	#5	#4	#3	#2	#1	#0
9000								ACAN

『Way of Validating』: After power-on

『Default Setting』: 0000 0000

#0 ACAN: GSK-CANA function on all system servo is

0: Not used

1: Used

9010	Communication baud rate of GSK-CANA function on all system servo(ABPS)

『Way of Validating』: After power-on

『Default Setting』: 500 (kbps)

『Value Range』; 500, 600, 800 or 1000 (kbps)

ABPS Set communication baud rate of GSK-CANA function on system servo.

Note: The baud rate set by the servo drive unit parameter should be consistent with the one set by the parameter.

9011	Slave number corresponding to each axis during servo communication（SIDx）

『Way of Validating』: After power-on

『Value Range』: 0~5

『Default Setting』: 0

SIDx The parameter sets the slave number corresponding to each axis during servo communication.

Note: "0" represents the axis doesn't connect with the servo subunit. "1~5" represent the servo slave number corresponding to each axis.

9012	**Slave number corresponding to the extended servo spindle communication（SIDS1）**

『Way of Validating』: After power-on

『Default Setting』: 0

『Value Range』: 0~5

The parameter sets the corresponding slave number during the servo spindle communication when the total controlled axes exceed the range.

Note: "0" represents the axis doesn't connect with the servo subunit. "1~5" represent the analog spindle slave number corresponding to the axis.

Appendix 1 Alarm List

1.1 Program Alarms (P/S Alarms)

No.	Message	Contents
000	Emergency stop alarm, ESP input open circuit	Recover ESP signal input to clear alarm.
001	Part prog. open failure	Reset to clear alarm or power-on again.
002	Single block exceeds 256 characters	Characters excessive in single block; modify the program.
003	Data exceeds permissive range	Input data exceeds permissive range, or the specified data exceeds 8 digits; modify the data.
004	Address not found	With number or symbol other than address at the beginning of a block. Modify the program.
005	No data follows address	No data follows address or expression format following address checks error, without brackets. Modify the program.
006	Illegal use of negative sign	Sign "-" was input after an address with which it can't be used, or two or more "-" was input. Modify the program.
007	Illegal use of decimal point	Decimal point"." was input after an address with which it can't be use, or two or more "." was input. Modify the program.
008	Input illegal address	Input unusable address in significant area. Modify the program.
009	Incorrect G code	Specify improper G code or that with functions not provided. Modify the program.
010	Address repetition error	Specify the same address twice or more in a block, or specify two or more G codes in same group in a block. Refer to para. 3403#6 AD2. Modify the program.
011	Command can't run in MDI mode	Command cannot run in MDI mode. Modify the program.
012	Too many M codes	Multiple M codes can't be specified in a block, referring to para.3404#7 M3B. Modify the program.
014	Divided by zero	Divisor was 0(including tan90°). Modify the program.
017	Para. modified failure.	Check that the para. file be abnormal. User partition is possibly damaged !
018	Part prog. operation failure	Reset to clear alarm.
019	End of record	Specify end symbol (%) of record, or not specify end of program, referring to para.3404#6 EOR. Modify the program.
020	DNC time out	DNC transmission failure; Please check.
021	Feedrate out of range	Feedrate was not commanded to a cutting feed or the feedrate was inadquate. The meaning of F is determined by G98/G99, please check current modal

No.	Message	Contents
		of G98/G99. Modify the program.
022	Spindle speed out of range	Improper spindle speed or spindle surface speed value, referring to para.3031 SCB. Modify the program.
023	Number followed M code out of range	Specify undefined M code, referring to para.3030 MCB. Modify the program.
024	Improper G code	The G code can't in the same block with other G codes. Modify the program.
025	Illegal tool No.	Specify a tool No. which doesn't exist, referring to para. 3032 TCB. Modify the program.
026	Illegal offset No.	Tool offset No. too large selected by T code. Modify the program.
027	Illegal offset value	Tool offset value selected by T code too large. Modify the program.
028	T code not allowed in the block	Can't specify T code in a block in which G50,G10 and G04 exists, referring to para.5006#1 TGC. Modify the program.
031	Too many axes commanded	Attempt was made to move the tool along more than maximum number of simultaneously controlled axes. Modify the program.
032	Illegal axis for interpolation	An axis not included in selected level commanded in interpolation command, or basic axis with its parallel axis were commanded simultaneously that impossible to interpolate. Modify the program.
033	Illegal level axis commanded	An axis not included in selected level commanded in circular interpolation. Modify the program.
034	No radius commanded	In circular interpolation, R,I,J,K has not been specified, referring to para. 3403#5 CIR. Modify the program.
035	Illegal radius	In circular interpolation, R specifies incorrect value, referring to para.3403#4 RER. Modify the program.
036	Over tolerance of radius	In circular interpolation, difference of the distance between start point and the center of an arc and that between end point and the center of an arc exceeded setting value, referring to para.3410 CRE. Modify the program.
037	Chamfering amount, J, K, was specified error in thread cutting commands	Chamfering amount exceeds permissive range. The number followed K is less than zero in G32, G34. The number followed J or K is less than zero in G92. Modify the program.
038	Illegal lead command	Lead specified by F is out of range, or in variable threading, the lead incremental and decremental specified by R exceeded permissive range. Modify the program.
039	Chamfering amount too large of long axis in threading.	Chamfering amount of long axis was greater than thread length. Modify the program.

No.	Message	Contents
040	Chamfering amount too large of latitude axis in threading.	Chamfering amount of latitude axis in G92 was greater than the distance between start point and end point. Modify the program.
041	Illegal level select	In the level selection command, two or more axes in the same direction are selected. Modify the program.
042	Metric/inch conversion command error	Metric/inch conversion code G20/G21 was not specified in an independent block at the beginning of the program or execute metric/inch conversion in subprogram call. Modify the program.
043	Reference return incomplete	Reference return can't be performed normally because the start point is too close to the reference position or the speed is too slow. Separate the start point far enough away from the reference position. Or specify a sufficiently fast speed for reference point return.
044	Reference return incomplete	In auto operation halt state, manual reference point return can't be performed.
045	The axis is not at reference point	The axis does not return to reference point in G27. Check the program content.
046	G28 found in sequence return	A command of program restart was specified without the reference position return operation after power-on or emergency stop, G28 was found during search. Perform the reference position return.
047	The axis does not turn to reference point	Didn't return to reference point before cycle start. Perform the reference return first.
048	Illegal reference point	Address P specifies other values than 2~4 in G30.\nModify the program.
051	G37 arrival signal not asserted	In auto tool compensation function (G36,G37),within the area specified by parameter. Measurement position reach signal (XAE or EAE) is not turned on. This is due to a setting or operator error.
052	Offset number not found in G37	Auto tool compensation (G36\G37) was specified without T code. Modify the program.
053	T code not allowed in G37	T code and auto tool compensation (G36, G37) was specified in the same block. Modify the program.
054	Illegal axis command in G37	In auto tool compensation function (G36,G37),an invalid axis is specified or the command is incremental. Modify the program.
055	G37 function is disabled	Auto tool compensation function is disabled (G36, G37) with reference to PARA.6240#7 IGA. Modify the program.
058	G31 not allowed in G99.	G31 skip cutting is commanded in the per revolution. Modify the program
059	G31 not allowed in tool radius compensation mode.	In tool nose radius compensation mode, specify skip cutting command. Modify the program.
061	Illegal P command in G10	In setting an offset amount by G10, the offset number

No.	Message	Contents
		is excessive or not specified. Modify the program.
062	Illegal offset value G10	In setting an offset amount by G10, the offset value specified by P is excessive or not specified. Modify the program.
063	Format error in G10 or L50	Any of the following occurs at the programmable-parameter input：Address N or R was not entered. A number not specified for a parameter was entered. The axis No. is too large. The axis number was not specified in the axis-type parameter. An axis was specified in the parameter which is not an axis type. Modify the program
065	Cumulated shift out of range	The result of calculation is out of allowable range. Modify the program.
068	A stroke limit check inhibited area error	The coordinate for para of the plus side inhibited area is not greater than that of the minus side inhibited area, or the difference is not greater than 2000 output increment, referring to para No.1322 & No.1323\nModify the program.
071	Spindle orientation please	Without any spindle orientation, an attempt was made for spindle indexing. Perform spindle orientation.
072	C/H code and move cmd in same block	A move command of other axis was specified in same block as spindle indexing address C, H. Modify the program.
073	M code and move cmd in same block	A move command of other axis was specified in same block as spindle indexing address M. Modify the program.
074	Illegal command G12.1/ G13.1	The conditions are incorrect when polar interpolation is started or cancelled.\n1)In modes other than G40, G12.1/G13.1 was specified.\n2)An error is found in level selection. Para assignment incorrect. Modify the program.
075	Improper G code	Specify G code which can't be used in polar coordinate interpolation. Modify the program.
081	Address P not defined	Address P(program number) was not commanded in block including M98, G65 or G66. Modify the program.
082	Subprogram nesting error	The subprogram call exceeds 12 folds. Modify the program.
083	Program number not found	The program number was not found specified by P in M98, M99, G65 or G66. Modify the program.
084	Subprogram call error	A program can't call main program or itself in M98,G65 or G66. Modify the program.
085	Program call statement can't run in MDI&DNC operation	Marco program and subprogram call in MDI &DNC operation isn't supported. Modify the program.
090	Axis specified error in constant surface speed	In G96 modal, the specified axis by parameter is wrong. Modify the parameter.

No.	Message	Contents
	control	
101	Over-speed of spindle in threading	In threading, the spindle speed specified is too fast for the threading axis. Modify the program.
121	Canned cycle cmd in non ZX level	Canned cycle can't command in non ZX level. Modify the program.
122	Specify other axes not included in ZX level.	Specify other axes not included in ZX level. Modify the program.
123	The R value (radius value) is greater than the U value (absolute value) in G90, G92 commands.	Absolute values of R is greater than that of U in G90,G92 while their signs are inconsistent. Modify the program.
124	Absolute values of R is greater than W in G94.	Absolute values of R is greater than that of W in G94 while their signs are inconsistent. Modify the program.
126	Illegal level select in multiple repetitive cycle	Multiple repetitive cycle was commanded in non ZX level. Modify the program.
127	Specify other axes not included in ZX level in G70~G76	Specify other axes not included in ZX level in G70~G76 and move command between ns-nf. Modify the program.
128	Illegal G code in G70~G73	Specify unusable G code between ns-nf specified by P & Q . Modify the program.
129	G70~G73 cannot operate in MDI mode	G70~G73 with P & Q was specified in MDI mode. Modify the program.
130	Illegal macro statement in G70~G73.	Macro statement is unallowable in G70~G73 command. Modify the program.
131	Illegal subprogram call in G70~G73	Subprogram call is unallowable in the end move command specified by P & Q in G70~G73\nModify the program.
132	Illegal subprogram call in G70~G73	Subprogram call is unallowable in G70~G73 command. Modify the program.
133	P or Q is out of range in G70~G73	P & Q was not commanded or out of range in G70~G73 command. Modify the program.
134	Sequence number not found in G70~G73	The sequence number specified by P & Q was not found in G70,G71,G72 or G73. Modify the program.
135	Number followed P, Q error in G70~G73	The number specified by address P & Q the same in G70~G73.\nModify the program.
136	The two blocks as components of G71~G73 command are discontinuous	The two blocks as components of G71~G73 command are discontinuous, which is possible to cause error. Modify the program.
137	Blocks between Ns & Nf exceeds 100 in G70~G73	Too many blocks between Ns & Nf blocks in G70~G73. Modify the program.
138	Target shape between Ns & Nf is not monotonous in G71~G73 command	A target shape which is not monotonous increase or decrease is specified in multiple repetitive cycle(G71 or G72), or in G73 Z axis isn't monotonous, or X axis isn't monotonous while there is chamfering or finishing

No.	Message	Contents
		allowance along Z axis, referring to para.5102#1 MRC. Modify the program."
139	Start point was on cutting path in G71~G73	Start point was on cutting path in G71~G73, which may cause interfere of tool and workpiece, referring to para. 5104#2 FCK. Modify the program.
141	Direction of chamfering and finishing allowance along X axis is inconsistent in G73	Direction of chamfering and finishing allowance along X axis is inconsistent in G73.\nModify the program.
142	Direction of chamfering and finishing allowance along Z axis is inconsistent in G73	Direction of chamfering and finishing allowance along Z axis is inconsistent in G73.\nModify the program.
143	Finishing allowance in G70~G73 out of range	Finishing allowance in G70~G73 is out of range. Modify the program.
144	G00 or G01 move command not found in first block of G71~G72	G00 or G01 move command should include in first block of G71~G72.\nModify the program.
145	G00-G03 move command not found in first block of G73	G00-G03 move command not found in first block of G73.\nModify the program.
146	X axis motion in the first block of G71	X axis increment was not commanded in first block of G71, or X axis increment is zero, or Z axis increment was commanded. Modify the program.
147	Z axis motion in the first block of G72	Z axis increment was not commanded in first block of G72, or Z axis increment is zero, or X axis increment was commanded. Modify the program.
148	Depth of cutting is less than zero in G71 or G72	Escaping amount is less than zero in G71 or G72. Modify the program.
149	Escaping amount is less than zero in G71 or G72	Escaping amount is less than zero in G71 or G72. Modify the program.
150	Increment cutting amount out of range in G73	Increment cutting amount out of range in G73. Modify the program.
151	The number of division R(d) in G73 out of range	The rounding number of division is less than 1 or more than 999. Modify the program.
152	Z axis increment not specified in G74	Z axis increment was not specified in G74. Modify the program.
153	Number followed address Q is out of range in G74	Number followed address Q is out of range in G74. Modify the program.
154	X axis increment not specified in G74	X axis increment was not specified in G75. Modify the program.
155	Number followed address P is out of range in G74	Number followed address P is out of range in G74. Modify the program.
156	R(e) is less than zero in G74 or G75	Return amount R(e) is less than zero in G74 or G75. Modify the program.
157	R(△d) is less than zero in G74 or G75	Relief amount of tool at cutting bottom R(△d) is less than zero. Modify the program.

No.	Message	Contents
158	Depth of cut in G74 or G75 out of range	Depth of cut in X or Z direction in G74 or G75 is out of range. Modify the program.
160	,X or Z axis increment is 0 in G76	X or Z axis increment is 0 in G76. Modify the program.
161	Repetitive count in finishing is less than 1 or greater than 99 in G76	Repetitive count in finishing is less than 1 or greater than 99 in G76.\nModify the program.
162	Chamfering amount out of range in G76	Angle of tool tip out of range in G76. Modify the program.
163	Q(△dmin) out of range in G76	Minimum cutting depth Q(△dmin) out of range in G76. Modify the program.
164	In G76, Q(△dmin) exceeds the permitted range	Minimum cutting depth Q(△dmin) out of range in G76. Modify the program.
165	Finishing allowance R(d) out of range in G76	Finishing allowance R(d) is less than least increment in G76. Modify the program.
166	R and U is inconsistent for taper thread cutting in G76	Machining start position is between thread beginning point and end point in G76. Modify the program.
167	Thread height not specified by P in G76	Thread height not specified by P in G76.\nModify the program.
168	Thread height is less than Finishing allowance or minimum cutting depth in G76	Thread height is less than Finishing allowance or minimum cutting depth in G76. Modify the program.
169	Number followed address Q is out of range in G76	Depth of cut in 1st cut Q was out of range, or not specified. Modify the program.
180	Illegal S code command in rigid tapping	In rigid tapping, an S value is out of range or not specified. Modify the program.
181	Illegal K in tapping	Repetitive count, K, in rigid tapping is out of range. Modify the program.
182	Illegal F in tapping	Feedrate specified is out of range in tapping. Check G98 & G99 modal. Modify the program.
183	Program miss at rigid tapping	Position for rigid M code and S command is incorrect in rigid tapping. Modify the program.
184	Illegal axis operation in rigid tapping	In rigid tapping an axis movement is specified between the rigid M code block and G84 block. Modify the program.
185	The spindle of rigid taping is not selected	The spindle of rigid taping is not selected or the specified axis cannot be used to tapping. Modify the program.
186	Level changes while tapping	Non G18 level was selected when tapping or start tapping in non G18 level. Modify the program.
187	Incorrect data in tapping	The specified distance is too short or long in tapping. Modify the program.
188	Unusable data specified in	Specify other M code or S code between rigid tapping

No.	Message	Contents
	tapping	M code block and G84 block. Modify the program.
189	Unallowed M code in rigid tapping	In rigid tapping, rigid tapping M code can't be in the same block with M code for Cs-axis clamping in canned cycle for drilling. Modify the program.
190	servo spindle positioning cmd in rigid tapping	Servo spindle increment in positioning cmd was specified in rigid tapping. Modify the program.
197	C-axis commanded in spindle mode	The program specified a movement along the Cs-axis when the signal CON(G27#7) is OFF. Correct the program, or consult PLC program to find the reason the signal is not turned on.
198	Spindle speed reached signal not detected	Spindle speed reached signal(SAR) was not detected enabled when cutting. Modify the program or check the PLC.
201	Improper command in custom macro program	A function which can't be used in custom macro program is commanded. Modify the program.
202	Format error in macro program	There is an error in other format than <Formula>. Modify the program.
203	Illegal variable number in macro program	A value not defined as a variable number is designated in the custom macro. Modify the program.
204	Unallowable macro program call	A program in G66 modal specified M98, G65 or G66. Modify the program.
205	The nesting of bracket exceeds the upper limit	The nesting of bracket exceeds the upper limit(quintuple). Modify the program.
206	Illegal argument	The SQRT argument is negative, or BIN argument is negative, or other values other 0~9 are present on each line of BIN argument. Modify the program.
207	Quadruple macro modal call	A total of four macro call and macro modal calls are nested. Modify the program.
208	Macro control command cannot be used in DNC and MDI program	Macro control command was specified in DNC and MDI mode. Modify the program.
209	Missing end statement	DO-END does not correspond to 1：1.\nOr has other illegal cmd exists in END block, incorrect format. Or control jumped into loop. Modify the program.
210	Substution statement of custom macro not allowed	User's authority is too low to execute subtution statement of custom macro. Modify the program.
211	Illegal loop number	In DOn, 1≤n≤3 is not established. Modify the program.
212	NC and macro statement in same block	NC and custom macro coexist.\ Modify the program.
213	Illegal macro sequence number	The sequence number specified in the branch statement was not 1~99999, or, it can't be searched. Modify the program.
214	Illegal argument address	An unallowable argument address was used which is not in <Argument Designation>. Modify the program.
216	Illegal argument	The argument is incorrect, or the argument is illegal.

No.	Message	Contents
		Modify the program..
217	Operand of logical operation statement error	Operand of logical operation statement OR,XOR,AND are negative. Modify the program.
218	G67 (custom macro cancel)	G67 was commanded while corresponding G66 command was not found. Please check the program whether G66 should be added, and referring to para 6000#1 G67.\nModify the program.
231	The same axis was commanded by PLC and CNC	Axis control command was given by PLC to an axis controlled by CNC. Modify the program or check the PLC.
232	Cannot change PLC control mode	Select an axis which is in commanding by PLC control. Modify the PLC program.
251	No solution at NRC	A point of intersection can't determined for tool nose radius compensation. Modify the program.
252	Not allowed to start & cancel NRC in arc command	Start or cancel tool nose radius compensation in circular interpolation. Modify the program.
253	Can't change level in NRC	The offset level is switched in tool nose radius compensation. Modify the program.
254	Interference in circular block	The arc start point or end point coincides with arc center. Overcut will occur in tool nose radius compensation. Modify the program.
255	Interference in G90 or G94 block	Overcut will occur in tool nose radius compensation in canned cycle G90 and G94. Modify the program.
256	Interference in arc concluded from checking	Overcut is possible to occur in tool nose radius compensation. Modify the program.
257	Inconsistent of direction of tool path in NRC and on drawing	Inconsistent of direction of tool path in NRC and on drawing(if exceeds range between 90 and 270 degree)possibly result in part overcut. Modify the program.
258	G41 or G42 not allowed in MDI mode	G41 or G42 was specified in MDI mode(tool nose radius compensation),referring to para 5008#4 MCR. Modify the program.
259	Inner whole circle cutting overcut	In inner whole circle cutting, overcut possibly occur, referring to para 5008#5 CNF. Modify the program.
260	Undercut in machining step being less than tool radius	undercut in machining step being less than tool radius, search 5008#6 CNS. Modify the program.
261	Radius of arc is less than that of tool in inner surface arc cutting	Radius of arc is less than that of tool in inner surface arc cutting, which might cause overcut. Modify the program.
262	Arc cmd exists when cancel temporarily or create NRC	While NRC is canceled temporarily as a result of a non-NRC G code, an arc command was specified. Modify the program.
263	NRC detected error	Detect error in tool nose radius compensation. This is due to program or operator . Modify the program.
281	Illegal tool group number	Tool group number exceeds maximum allowable value.

No.	Message	Contents
		Modify the program.
282	Tool group number not found	Tool group number commanded in machining program is not set. Modify the program or parameter.
283	No space for tool entry	The number of tools within one group exceeds the maximum value registerable. Modify tool number.
284	T code not found	In tool life registeration, a T code was not specified where is should be. Modify the program.
285	P/L command not found	P/L commands are missing at the head of program in which the tool group is set. Modify the program.
286	Too many tool groups	The number of tool groups to be set exceeds maximum allowable value. Modify the program.
287	Illegal tool life data	The tool life to be set is too excessive. Modify the setting value.
288	Tool data setting incompleted	During executing a life data setting program, power was turned off. Set again.

1.2 Parameter Alarms

No.	Message	Contents
400	Parameter switch is ON	Press 【RESET】key to cancel the alarm.
401	Duplicated servo id was set for control-axis	Modify para.No.9020.
402	Parameters back up failure	Check the memory or power-on again.
403	Parameters recover failure.	Check whether parameters are being written in, or power-on and retry.
404	Duplicated axis name were set	Modify para.No.1020.
406	Servo comm id the same between none-Cs axis and spindle	Modify para. No. 3704, No.8133, No.9020, No.9030.
407	Servo comm id inconsistent between the Cs-axis and spindle	Modify para.No.3704,No.8133, No.9020, No.9030.
408	Servo comm id the same between different spindle	Modify para. No. 9030.
450	Please turn off the power	A parameter which requires the power off was input, turn off power.
452	Number of CNC controllable axes exceeds the total number	Check para. No.1010 and 8130.

No.	Message	Contents
453	Duplicated axis attribution were set	Modify para. No. 1022.
454	Duplicated servo control No. were set	Modify para. No. 1023.
455	Attribution of rotary axis error	Para No.1006 and No.1022 conflict, and axis attribution of rotary axis cannot be 0. Modify para No.1006 or No.1022.

1.3 Pulse Encoder Alarms

No.	Message	Contents
500	Return to reference position	Manual reference return required.
501	Absolute pulse encoder alarm: communication failure	Absolute pulse encoder communication error. Data transmission error. Reasons include: pulse encoder error, cable or servo interface module failure.
502	Absolute pulse encoder alarm: overtime error	Absolute pulse encoder overtime error. Data transmission error. Reasons include: pulse encoder error, cable or servo interface module failure.

1.4 Servo Alarms

No.	Message	Contents
604	Servo alarm	digital servo unit detect fault. Check the servo or modify para. No.1816
650	Power supply to the servo is turned off	The coordinate system became inacurrate when the control command to the servo is interrupted. Please return to the reference position..

1.5 Overtravel Alarms

No.	Message	Contents
700	Stored stroke limit1: +	Exceeded the + sides stored stroke limit 1. Modify para No.1320 or No.1326.
701	Stored stroke limit1: -	Exceeded the - sides stored stroke limit 1. Modify No.1321 or No.1327.
702	Stored stroke limit2: +	Exceeded the + sides stored stroke limit 2. Modify para No.1322.
703	Stored stroke limit2: -	Exceeded the - sides stored stroke limit 2. modify No.1323.
704	Stored stroke limit3: +	Exceeded the + sides stored stroke limit 3. Modify para No.1324.
705	Stored stroke limit3: -	Exceeded the - sides stored stroke limit 3. modify No.1325.

No.	Message	Contents
706	Over travel: +	Exceeds + side overtravel limit. Press 【overtravel cancel】and manual exit overtravel area or modify para No.3004.
707	Overtravel : -	Exceeds - side overtravel limit. Press 【overtravel cancel】and manual exit overtravel area or modify para No.3004.

1.6 Spindle Alarms

No.	Message	Contents
800	Spindle 1 alarm	Spindle 1 alarms.
810	Spindle 2 alarm	Spindle 2 alarms.

1.7 System Alarms

No.	Message	Contents
900	Memory alarm	Storage allocation error.
909	TRYOUT timed out. System functions are restricted.	Please contact the dealer.
910	Initial parameter failure	User parameter file does not exist or data is damaged. Default parameters become effective.
911	Initial CNC configuration failure	CNC config file does not exist or data is damaged. Default configuration becomes effective.
912	Initial tool offset data failure	Tool offset file does not exist or data is damaged. Initial data becomes effective.
913	Initial tool life data failure	Tool life file does not exist or data is damaged. Initial data becomes effective.
914	Initial pitch error compensation data failure	Pitch error compensation file does not exist or data is damaged. Initial data becomes effective.
915	Initial PLC programe failure	Read file failure in registering program, or compile failure.
916	CNC Initialization failure	Power on again.
917	GSK-CAN initialization failure	Please check: (1) Whether the communication baud-rate is consistent between CNC and servo(Cnc para. Nos.9010, 9011, & corresponding servo para.) (2)Whether the communication servo-id is consistent between CNC and servo (Cnc para.Nos.9020, 9030, & corresponding servo para.) (3)Whether the communication cable is connected well, grounding is abnormal, and the terminal resistance is installed. Please power on again.

918	Keyboard on MDI panel or operator's panel failure	Press【RESET】to cancel alarm, or power-on again.
919	Memory failure, which needs repair, or power-on again	Press【RESET】to cancel alarm and power-on again, or refer to manufacturer for repair.
920	Too many alarm and info	The number of alarm exceeds 14 or number of info exceeds 20.
921	Undefined alarm No.	Missing alarm content for alarm No..
922	Format error in alarm content	Part of data in alarm content and operation info was incorrect.
950	Pulse error FPGA received from system	FPGA received pulse does not coincide with that system has sent.
998	Data abort	Please contact us.

1.8 Communication prompt on the operation panel

No.	**Message**	**Contents**
4200	Communication on the panel error	Communication between the panel and the system is mistaken. Please check the connection between them.
4201	Edit keyboard error	Edit keyboard input is mistaken. Please restart the system.

1.9 GSK-CAN Communication Prompts

No.	**Message**	**Contents**	**Possible Reason**
5000	GSK-CAN slave configuration method error	GSK-CAN extended function unusable.	Unused at present
5001	I/O unit missing in GSK-CAN communication	The IO unit control function is unusable.	Unused at present
5002	Extended slave is missing in GSK-CAN communication	The extended axis function is unusable.	Unused at present
5003	communication error	Please check whether the communication interface is loose, the power supply is grounded properly, or the end resistance is installed, then, turn on the power.	During GSK-CAN communication, if error continuously occurs in all slaves, this prompt is displayed.
5004	GSK-CAN slave ID number	Modify the parameter for slave number and re-power	This prompt is displayed when two slave numbers of servos are set the

No.	Message	Contents	Possible Reason
	conflicted	on (cut off the GSK-CAN connection before parameter modification at the server side)	same.
5005	All GSK-CAN slaves connections failure	Check the setting of parameter No.9000-No.9012 and check whether the communication interface is loose, the power supply is grounded properly or the end resistance is installed, then turn ON the power again.	When GSK-CAN is restarted or re-connected, all the slaves are cut OFF. The possible reasons are: (1). Poor contact of system GSK-CAN communication interface (2). Poor contact of servo slave GSK-CAN communication interface. (3). End resistor is not installed on the servo slave which is the farthest from the system. (4) GSK-CAN communication is interrupted. (5). Power supply is not grounded.
5006	n-th axis GSK-CAN slave connection failure	Check whether the communication interface is loose or the power supply is grounded properly.	The same as the prompt No. 5005, but this prompt indicates that only some slave connection is failed.
5010	n-th axis servo model and software version read failure	Check whether the communication interface is loose or the power supply is grounded properly, then turn ON the power again.	GSK-CAN communication is interrupted.
5011	n-th axis servo configuration failure	Please update relevant servo configuration file and turn ON the power again.	The servo configuration file is not found, or the data in the file is unusable.
5020	n-th servo parameter read failure	Please check whether the communication interface is loose or the power supply is grounded	GSK-CAN communication is interrupted
5030	The parameter in the n-th axis current servo parameter file is inconsistent with the read one	Please select a valid servo parameter.	After the servo is disconnected with the system, servo parameter is manually changed on the drive unit. When this servo is used the next time after power-on, an alarm occurs. Note: When a servo of different version is used, the system will automatically select the parameters read in the servo, and an alarm will not occur.

No.	Message	Contents	Possible Reason
5031	The parameter of the n-th production servo parameter is inconsistent with the read one	The parameter of the n-th production servo parameter is inconsistent with the read one (such as the encoder zero drift, drive unit version etc.) You could select the read servo parameter or other parameter stored in CNC servo parameter files.	This alarm occurs together with alarm No. 5030; It occurs only when some parameters (such as encoder zero drift, drive unit version) are inconsistent with the current stored parameters. This parameter includes two types: one can be modified manually after the communication is disconnected and logining in the drive unit; the other one can only be modified by upgrading servo software. When the two types are not consistent, the parameter read from the servo system should take priority and the stored value in CNC current parameter file should be overwritten.

1.10 Servo Inner Alarms

Note: (1) n represents the sequence number of GSK-CAN servo slaves set by system parameters (ranges from 1~9).

(2) The examples shown in the following table are feed servo V1.03 and spindle V2.02. Previous versions are compatible.

(3) The following content is valid till this user manual is issued and it is changed without further notice. Please refer to the latest servo manual.

Feed Servo	DAT2030C, DAT2050C, DAT2075C, DAT2100C (V1.03 or the earlier version)	
No.	Message	Contents
5n00	Normal	
5n01	Overspeed	The speed of servo motor exceeds the setting value.
5n02	Overvoltage	The main voltage is too high.
5n03	Undervoltage	The main voltage is too low.
5n04	Excess position deviation	The position deviation value exceeds the setting value.
5n05	Overheat	The temperature of the motor is too high.
5n06	Speed amplifier saturated	The speed regulator is saturated for a long time.
5n07	Drive unit inhabitation abnormal	The drive unit input inhabitation is OFF.
5n08	Position deviation counter overflow	The absolute value of position deviation counter value exceeds $2^{30.}$
5n09	Coder fault	Coder signal error
5n10	Undervoltage of control	The voltage of the control power is less than

	power	±15V.
5n11	IPM module fault	IPM intelligent module fault
5n12	Overcurrent	The current of the motor is excessive.
5n13	Unused	
5n14	Braking fault	Braking circuit fault
5n15	Unused	
5n16	Motor overheat	The heat value of the motor exceeds the setting value. (I^2t detection)
5n17	Unused	
5n18	Unused	
5n19	Unused	
5n20	EEPROM error	(EEPROM) error
5n21	Phase lose alarm	Phase lose during the three-phase AC current input
5n22	Coder zeroing alarm	The encoder cannot perform normal regulation.
5n23	Current sampling circuit fault	A/D chip or current sensor error
5n24	Unused	
5n25	Unused	
5n27	Unused	
5n28	Software upgrade prompt alarm	The alarm is issued when the system software is upgrading.
5n29	Parameter error	The parameter is out of the controllable range.
5n30	Unused	
5n31	Unused	
5n32	illegal code in UVW signal	Full high-level or full low-level exists in UVW signal.
5n33	Power charging fault	Charging circuit is damaged.
5n34	Pulse electronic gear ratio is excessive	The parameter of pulse electronic gear ratio is incorrect.
5n35	No external connected brake pipe	There is no external connected brake pipe or the pipe is faulty.
5n36	Three-phase power OFF	Three-phase power OFF or three-phase power detection circuit is faulty.
5n37	The temperature of the radiator is too low	
5n38	The temperature of the radiator is too high	
5n39	Absolute encoder single-ring read alarm	
5n40	Absolute encoder multi-ring read alarm	
5n41	Encoder type configuration error	The encoder type set by drive unit is inconsistent with the encoder type of the motor.
5n42	EEPROM alarm in absolute encoder	

5n43	EEPROM check error in absolute encoder	
5n44	Coder type error	Please check parameter No. PA97.
5n45	Data check error in absolute encoder	Data check error in sensor mode.

Spindle Servo	**DAY3025C, DAY3100C, DAP03C (V2.02)**	
No.	**Message**	**Contents**
5n00	Normal	
5n01	Motor overspeed	The speed of the spindle motor exceeds the setting value.
5n02	Main circuit overvoltage	The voltage of the main circuit power is excessive.
5n03	Main circuit undervoltage	The voltage of the main circuit power is too low.
5n04	Excess position deviation	The position deviation value exceeds the setting value.
5n05	Motor overheat	The temperature of the motor is too high.
5n06	Unused	
5n07	Unused	
5n08	Position deviation counter overflow	The absolute value of position deviation counter value exceeds 2^30.
5n09	Motor encoder fault	The signal of motor encoder is faulty.
5n10	Unused	
5n11	IPM module fault	IPM intelligent module fault
5n12	Unused	
5n13	Overload	The current of the motor is excessive.
5n14	Unused	
5n15	Unused	
5n16	Motor overheat	The spindle servo drive unit and motor are overloaded (temporary overheat).
5n17	Excess braking time	This alarm is issued when the discharging time is too long.
5n18	Braking circuit fault 1	No braking signal, no braking feedback
5n19	Braking circuit fault 2	No braking signal, no braking feedback
5n20	EEPROM error	EEPROM error
5n21	Phase lose alarm	At least one of the R, S, T of three-phase power is off.
5n22	Unused	
5n23	Excessive current error	The zero drift is excessive.
5n24	Spindle encoding disc fault	The spindle encoder signal error
5n25	Orientation failure	The position cannot be found.
5n26	Cooling fins overheated	The cooling fins are overheated.
5n27	U, V, W connection error	The three-phase (U, V, W) sequence is wrong

5n28	The parameters are not re-adjusted or stored after upgrading	
5n29	The parameter value detected after power-on is out of the range	
5n30	Communication error	The connection between servo and CNC is faulty.
5n31	Unused	
5n32	Unused	
5n33	Charging alarm fault	The input voltage is less than 304V (DC bus voltage 430V).
5n34	Abnormal thermistor status	TEP-OH (TEM higher than 90°) or TEP-OL(TEP lower than -30°), the thermistor is short-circuited or cut off.

Appendix 2 Standard Ladder Function Allocation

2.1 X, Y Addresses Definition

Caution:

The general I/O signal (except those signals marked for fixed addresses) in GSK988T CNC system is defined by the embedded PLC (ladder diagram) program. When this CNC system is installed, the exact I/O functions are determined by the machine tool builder. Please refer to the manual from machine tool builder for details.

Pay attention that in this chapter, the functions of general I/O signal (i.e. X,Y addresses) are just described for GSK988T standard PLC program.

General I/O Interface on Machine Tool

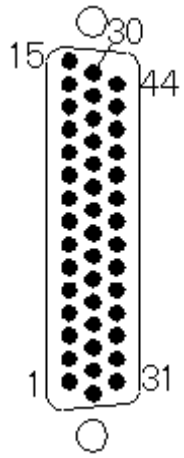


Fig. B-1 CN61 (male)input

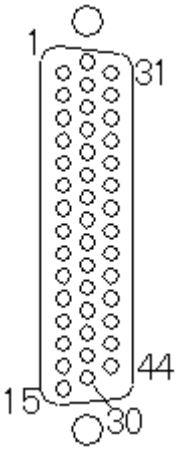


Fig. B-2 CN62 (female) output

DB Pin	PLC address	Function defined by standard PLC address		Remark
CN61.1	X0.0	SAGT	Protection door detection signal	
CN61.2	X0.1		Reserved	
CN61.3	X0.2	DIQP	Chuck input signal	
CN61.4	X0.3	DEC1	The 1st axis deceleration signal	Fixed address
CN61.5	X0.4	DITW	Tailstock control signal	
CN61.6	X0.5	ESP	Emergency stop input signal	Fixed address
CN61.7	X0.6	PRES	Pressure detection signal	
CN61.8	X0.7	T05	Tool position signal 5/ tool post pre-indexing signal (Yantai AK31)/Sensor E (Liuxin Tool Post)	
CN61.9	X1.0	T06	Tool position signal 6/ tool post pre-indexing signal (Yantai AK31)/Sensor F (Liuxin Tool Post)	
CN61.10	X1.1	T07	Tool position signal 7/ tool post overheat signal (Yantai AK31)	
CN61.11	X1.2	T08	Tool position signal 8	

DB Pin	PLC address	Function defined by standard PLC address		Remark
CN61.12	X1.3	DEC3	The 3rd axis deceleration signal	Fixed address
CN61.13	X1.4		Reserved	
CN61.14	X1.5	M41I	The 1st gear stage in-position	
CN61.15	X1.6	M42I	The 2nd gear stage in-position	
CN61.16	X1.7	T01	Tool position signal 1/T1 (Yantai AK31)/Sensor A (Liuxin Tool Post)	
CN61.29	X2.0	T02	Tool position signal 2/T2 (Yantai AK31)/ Sensor B (Liuxin Tool Post) Sensor A (Liuxin Tool Post)	
CN61.30	X2.1	T03	Tool position signal 3/T3 (Yantai AK31)/Sensor C (Liuxin Tool Post)	
CN61.31	X2.2	T04	Tool position signal 4/T4 (Yantai AK31)/Sensor D (Liuxin Tool Post)	
CN61.32	X2.3	DEC2	The 2nd axis deceleration signal	Fixed address
CN61.33	X2.4	DEC4	The 4th deceleration signal	Fixed address
CN61.34	X2.5	DEC5	The 5th deceleration signal	Fixed address
CN61.35	X2.6	TCP	Tool post lock signal Tool post proximity switch signal (Yantai AK31)	
CN61.36	X2.7	COIN	Spindle orientation completed signal	
CN61.37	X3.0	LMI1+	The 1st axis + side overtravel signal	
CN61.38	X3.1	LMI2+	The 2nd axis + side overtravel signal	
CN61.39	X3.2	LMI3+	The 3rd axis + side overtravel signal	
CN61.40	X3.3	WQPJ	Chuck in-position signal (outer chuck clamping and inner chuck unclamping)	
CN61.41	X3.4	NQPJ	Chuck in-position signal (inner chuck clamping and outer chuck unclamping)	
CN61.42	X3.5	SKIP	G31 skip signal	Fixed address
CN61.43	X3.6	G36	G36 skip signal	Fixed address
CN61.44	X3.7	G37	G37 skip signal	Fixed address
CN61.17	X4.0	LMI1-	The 1st axis – direction overtravel signal	
CN61.18	X4.1	LMI2-	The 2nd axis – direction overtravel signal	
CN61.19	X4.2	LMI3-	The 3rd axis – direction overtravel signal	
CN61.20	X4.3	LMI4+	The 4th axis + direction overtravel signal	
CN61.25	X4.4	LMI4-	The 4th axis - direction overtravel signal	
CN61.26	X4.5	LMI5+	The 5th axis + direction overtravel signal	
CN61.27	X4.6	LMI5-	The 5th axis - direction overtravel signal	

DB Pin	PLC address	Function defined by standard PLC address		Remark
CN61.28	X4.7		Reserved	
CN61.21 ～ CN61.24	0V			
CN62.1	Y0.0	M08	Cooling output signal	
CN62.2	Y0.1	M32	Lubrication output signal	
CN62.3	Y0.2		Reserved	
CN62.4	Y0.3	M03	Spindle CCW signal	
CN62.5	Y0.4	M04	Spindle CW signal	
CN62.6	Y0.5	M05	Spindle stop signal	
CN62.7	Y0.6		Reversed	
CN62.8	Y0.7	SPZD	Spindle braking output signal	
CN62.9	Y1.0	M41	Spindle gear 1 output signal	
CN62.10	Y1.1	M42	Spindle gear 2 output signal	
CN62.11	Y1.2	M43	Spindle gear 3 output signal	
CN62.12	Y1.3	M44	Spindle gear 4 output signal	
CN62.13	Y1.4	M12(DOQPJ)	Outer chuck clamping output / Inner chuck unclamping output signal	
CN62.14	Y1.5	M13(DOQPS)	Outer chuck unclamping output /inner chuck clamping output signal	
CN62.15	Y1.6	TL+	Tool post forward rotation output signal	
CN62.16	Y1.7	TL-	Tool post reverse rotation output signal	
CN62.29	Y2.0		Tool post motor braking signal (Yantai AK31)/ tool post unclamping output (Liuxin Tool Post)	
CN62.30	Y2.1		Tool post pre-indexing electromagnet signal (Yantai AK31)/ Tool post lock output (Liuxin Tool Post)	
CN62.31	Y2.2	YLAMP	Tri-colored lamp – yellow (normal state, non-running, non-alarm)	
CN62.32	Y2.3	GLAMP	Tri-colored lamp – green (running state)	
CN62.33	Y2.4	RLAMP	Tri-colored lamp – red (alarm state)	
CN62.34	Y2.5	M10	Tailstock advancing output signal	
CN62.35	Y2.6	M11	Tailstock retracting output signal	
CN62.36	Y2.7		Reserved	
CN62.37	Y3.0		Reserved	
CN62.38	Y3.1		Reserved	
CN62.39	Y3.2		Reserved	
CN62.40	Y3.3		Reserved	

DB Pin	PLC address	Function defined by standard PLC address		Remark
CN62.41	Y3.4	SORI	Spindle orientation signal	
CN62.42	Y3.5	SEC0	Spindle orientation selection signal 1	
CN62.43	Y3.6	SEC1	Spindle orientation selection signal 2	
CN62.44	Y3.7	SEC2	Spindle orientation selection signal 3	
CN62.17～CN62.19 CN62.26～CN6228			0V	
CN62.20～CN62.25			+24V	

Note1: Addresses X0.0~X0.7,X1.0~X1.7,X2.0~X2.7,X3.0~X3.7 are valid at a high-level, i.e. when the input signal +24V is connected, the state of address X signal is 1; when disconnected, the state is 0.

Note 2: When the state of address Y signal is 1, the output signal is connected to 0V (0V output); when the sate of address Y signal is 0, the output signal is at high-impedance state.

2.2 Standard Operation Panel

2.2.1 Address X

Address defined by PLC	Corresponding Key on the Panel	Remark
X18.0	Block skip	
X18.1	Auxiliary lock	
X18.2	Spindle override increase	
X18.3	Single block	
X18.4	Machine tool lock	
X18.5	Dry run	
X18.6	Spindle override decrease	
X18.7	Spindle override 100%	
X19.0	C axis moves along – direction(C -) /MPG C	
X19.1	C/S switch	
X19.2	Cycle start	
X19.3	Tailstock	
X19.4	The 4^{th} axis moves along – direction (4^{th} -)/MPG 4^{th}	
X19.5	Z axis moves along – direction (Z-)/ MPG Z	
X19.6	Y axis moves along – direction (Y-)/ MPG Y	

X19.7	X axis moves along – direction (X-)/ MPG X	
X20.0	Protection door	
X20.1	Tool post forward rotation	
X20.2	Tool offset	
X20.3	Tool post reverse rotation	
X20.4	Cooling	
X20.5	Spindle stop	
X20.6	Manual rapid traverse	
X20.7	Optional stop	
X21.0	Program restart	
X21.1	Spindle CW	
X21.2	Spindle jog	
X21.3	Spindle CCW	
X21.4	The 4th axis moves along + direction (4th+)	
X21.5	C axis moves along + direction (C+)	
X21.6	Spindle exact stop	
X21.7	Feed hold	
X22.0	MPG mode	
X22.1	Space key on the right of DNC	
X22.2	MANUAL mode	
X22.3	MDI mode	
X22.4	DNC mode	
X22.5	AUTO mode	
X22.6	REFERENCE POSITION RETURN mode	
X22.7	EDIT mode	
X23.0	Rapid traverse override 100%/MPG×1000	
X23.1	Z axis moves along + direction (Z+)	
X23.2	Rapid traverse 50%/ MPG×100	
X23.3	Rapid traverse 25%/ MPG×10	
X23.4	Y axis moves along + direction (Y+)	
X23.5	Rapid traverse F0/ MPG×1	
X23.6	X axis moves along + direction (X+)	
X23.7	Hydraulic pressure	
X24.0	Space key below the cycle start	
X24.1	Chuck	
X24.2	Lubrication	
X24.3	Space key on the right of spindle CCW	
X24.4 ～ X24.7	Undefined	System reserved

X25.0 ～ X25.7	Connected to terminal strip	Reserved for user
X26.0 ～ X26.7	Connected to terminal strip	Reserved for user
X27.0 ～ X27.7	Connected to terminal strip	Reserved for user
X28.0	Connected to terminal strip	Connected to panel baud switch (spindle override OV 1)
X28.1	Connected to terminal strip	Connected to panel baud switch (spindle override OV 2)
X28.2	Connected to terminal strip	Connected to panel baud switch (spindle override OV 3)
X28.3	Connected to terminal strip	Connected to panel baud switch (spindle override OV 4)
X28.4	Connected to terminal strip	Connected to panel baud switch (feedrate override OV1)
X28.5	Connected to terminal strip	Connected to panel baud switch (feedrate override OV2)
X28.6	Connected to terminal strip	Connected to panel baud switch (feedrate override OV3)
X28.7	Connected to terminal strip	Connected to panel baud switch (feedrate override OV4)
X29.0	Connected to terminal strip	Connected to panel button (cycle start)
X29.1	Connected to terminal strip	Connected to panel button (feed hold)
X29.2	Connected to terminal strip	Connected to panel key switch button (program protection lock)
X29.3	Connected to terminal strip	Connected to panel knob normally-open terminal (spindle rotation allowed)
X29.4	Connected to terminal strip	Connected to panel knob normally-closed terminal (feed allowed)
X29.5 ～ X29.7	Connected to terminal strip	Reserved for user

Note: The PLC address X18~X24 are the fixed addresses input by keys on the panel, and their functions are fixed. Addresses X25~X29 are lead to the terminal strip on the backboard of the panel, the exact functions are defined by the PLC run in the system.

2.2.2 Address Y

Address defined by PLC	Corresponding key on the panel	Remark
Y18.0	Block skip indicator	
Y18.1	Auxiliary lock key indicator	
Y18.2	L5 indicator	

Y18.3	Single block indicator	
Y18.4	Machine lock key indicator	
Y18.5	Dry run key indicator	
Y18.6	C/S switch key indicator	
Y18.7	C/S axis – direction key indicator	
Y19.0	C axis + direction (C+) key indictor	
Y19.1	The 4th axis + direction (4th+) key indicator	
Y19.2	Cycle start key indicator	
Y19.3	Feed hold key indicator	
Y19.4	Program restart key indicator	
Y19.5	Optional stop key indicator	
Y19.6	Spindle override decrease key indicator	
Y19.7	Spindle override 100% key indicator	
Y20.0	Spindle override increase indicator	
Y20.1	Hydraulic pressure key indicator	
Y20.2	Tailstock key indicator	
Y20.3	Lubrication key indicator	
Y20.4	Protection door key indicator	
Y20.5	Tool post forward rotation key indicator	
Y20.6	Tool offset key indicator	
Y20.7	Tool post reverse rotation key indicator	
Y21.0	Digitron (right) output (value 1)	
Y21.1	Digitron (right) output (value 2)	
Y21.2	Digitron (right) output (value 4)	
Y21.3	Digitron (right) output (value 8)	
Y21.4	Digitron (left) output (value 1)	
Y21.5	Digitron (left) output (value 2)	
Y21.6	Digitron (left) output (value 4)	
Y21.7	Digitron (left) output (value 8)	
Y22.0	MPG mode indicator	
Y22.1	Indicator of space key on the right of DNC	
Y22.2	MANUAL mode indicator	
Y22.3	MDI mode indicator	
Y22.4	DNC mode indicator	
Y22.5	AUTO mode indicator	
Y22.6	REF. mode indicator	
Y22.7	EDIT mode indicator	
Y23.0	Rapid traverse override 100% indicator	
Y23.1	Z axis + direction indicator	
Y23.2	Rapid traverse override 50% indicator	
Y23.3	Rapid traverse override 25% indicator	

Y23.4	Y axis + direction indicator	
Y23.5	Rapid traverse override F0 indicator	
Y23.6	X axis + direction indicator	
Y23.7	System alarm (ALM) indicator	
Y24.0	Cooling key indicator	
Y24.1	Chuck key indicator	
Y24.2	Indicator of space key on the right of the spindle CCW key	
Y24.3	Spindle exact stop key indicator	
Y24.4	Spindle stop key indicator	
Y24.5	Spindle CW key indicator	
Y24.6	Spindle jog key indicator	
Y24.7	Spindle CCW key indicator	
Y25.0	The 4^{th} – direction key indicator	
Y25.1	Z axis – direction (Z-) key indicator	
Y25.2	Y axis – direction (Y-) key indicator	
Y25.3	Z axis machine zero point indicator	
Y25.4	Y axis machine zero point indicator	
Y25.5	Z axis machine zero point indicator	
Y25.6	X axis – direction (X-) key indicator	
Y25.7	Rapid traverse key indicator	
Y26.0	Indicator of space key below the cycle start key	
Y26.1	L4 indicator	
Y26.2	L3 indicator	
Y26.3	L2 indicator	
Y26.4	L1 indicator	
Y26.5	System running (RUN) indicator	
Y26.6	C axis machine zero point indicator	
Y26.7	4^{th} axis machine zero point indicator	
Y27.0～Y27.7	Connected to terminal strip	Reserved for user
Y28.0～Y28.7	Connected to terminal strip	Reserved for user
Y29.0	Connected to terminal strip	Connected to panel button indicator (cycle start)
Y29.1	Connected to terminal strip	Connected to panel button indicator (feed hold)
Y29.2～Y29.7	Connected to terminal strip	Reserved for user

Note: The PLC addresses Y18~Y26 are the fixed addresses of indicator output on the panel; their functions fixed. Addresses Y27~Y29 are lead to the terminal strip on the backboard of the panel; the exact functions are defined by PLC.

2.3 Standard PLC Parameter Instruction

2.3.1 Parameter K

Note: K0~K7 do not need to be set.

Address	Parameter meaning	Initial value
K8.0	X axis manual movement direction (1: reversed, 0: not reversed)	0
K8.1	Y axis manual movement direction (1: reversed, 0: not reversed)	0
K8.2	Z axis manual movement direction (1: reversed, 0: not reversed)	0
K8.3	The 4th axis manual movement direction (1: reversed, 0: not reversed)	0
K8.4	C axis manual movement direction (1: reversed, 0: not reversed)	0
K9.0	Shield program protection lock (1: shield, 0: does not shield)	0
K9.7	Alarm occurs when invalid M code is commanded (1: yes, 0: no)	0
K10.0	Feed override (1: inversed, 0: not inversed)	0
K10.1	Turn off the spindle, cooling and lubrication output during reset (1: No, 0: Yes)	0
K10.2	Axes overtravel input signal alarm level (1:low-level alarm, 0: high-level alarm)	0
K10.3	Machine panel feed/spindle enable knob (1:valid, 0: invalid)	1
K10.4	Spindle type (1: gear, 0: analog)	0
K10.7	External emergency stop input signal (X0.5) (1: high-level alarm, 0: low-level alarm)	0
K11.0	Tool post lock signal (1: low-level, 0: high-level)	0
K11.1	Tool position signal (1: low-level, 0: high-level)	0
K11.2	Tool change method when standard tool change mode is selected (1: method A, 0: method B)	1
K11.3	Check tool position signal after tool change (1: Yes, 0: No)	0
K11.4	Check tool post lock signal (1: Yes, 0: No)	1
K11.6	Tool post selection (PB8 PB7: 00 standard tool post/01 Yantai Tool Post/10 Liuxin Tool post)	0
K11.7	Tool post selection (PB8 PB7: 00 standard tool post/01 Yantai Tool Post/10 Liuxin Tool post)	0
K12.0	1/0: manual inversed tool change is valid/invalid	0
K12.2	Zero return direction locked automatically (1: Yes, 0: No)	0
K12.5	Tri-colored lamp output function (1: enabled, 0: disabled)	0
K12.6	External hand-held unit (1: enabled, 0: disabled)	0
K12.7	Machine tool operation panel (1: MPU02B, 0: MPU02A)	0
K13.0	Chuck control function (1:enabled, 0:disabled)	1

K13.1	If the chuck function is valid, check the chuck clamping state when the spindle is started (1: Yes, 0: No)	1
K13.2	Tailstock control function (1: valid, 0: invalid)	0
K13.4	Spindle gear stage is stored when power-off (1: Yes, 0: No)	1
K13.5	Spindle automatic gear change in-position signal active level (1: low-level, 0: high-level)	0
K13.6	Check spindle automatic gear change in-position signal (1: Yes, 0: No)	0
K13.7	Spindle automatic gear change function (1: valid, 0: invalid)	0
K14.0	Check chuck clamping/unclamping signal (1:Yes, 0: No)	0
K14.2	Chuck mode (1: inner chuck, 0: outer chuck)	0
K14.4	Low-pressure alarm signal level (1: low-level alarm, 0: high-level alarm)	0
K14.5	Low-pressure alarm function (1: valid, 0: invalid)	0
K14.6	Protection door input signal alarm level (1: low-level alarm, 0: high-level alarm)	0
K14.7	Protection door alarm function (1: valid, 0: invalid)	0
K15.0	Starting up operation mode MD1	0
K15.1	Starting up operation mode MD2	0
K15.2	Starting up operation mode MD4	0
K15.4	Starting up operation mode (1: MD2, MD2, MD4, 0: the mode when power-off the last time)	0
K15.6	Servo spindle 8-point orientation function (1: valid, 0: invalid)	0

2.3.2 Parameter DT

DT address	PLC initial value	Minimum input value	Maximum input value	Meaning
DT0000	1000	0	60000	Spindle gear change time 1 (ms)
DT0001	1000	0	60000	Spindle gear change time 2 (ms)
DT0002	3000	0	60000	Low-pressure alarm detection time (ms)
DT0003	5000	100	5000	Tool change (for one tool position) time upper limit (ms)
DT0004	15000	1000	60000	Tool change (for maximum tool positions) time upper limit (ms)
DT0005	500	100	5000	M code execution duration (ms)
DT0006	500	100	5000	S code execution duration (ms)
DT0007	500	0	4000	Delay time of the tool post from forward rotation stop to reverse rotation output (ms)
DT0008	500	0	4000	Alarm time when the TCP signal is not received (ms)
DT0009	1000	0	4000	Tool post reverse rotation lock time (ms)
DT0010	0	0	10000	Delay time of M05 and spindle braking output (ms)

DT0011	50	0	60000	Spindle braking output time (ms)
DT0012	100	0	60000	Spindle jog time (ms)
DT0013	0	0	60000	Lubricating start time (0-60000ms) (0: no limit)
DT0016	0	0	60000	Automatic lubricating interval time (ms)
DT0017	0	0	60000	Automatic lubricating output time (ms)
DT0019	1000	100	60000	Chuck function execution duration when in-position signal is not checked (ms)
DT0021	1000	100	60000	Spindle stop, chuck operation enable delay time (ms)
DT0022	500	100	1000	Alarm indicator flickering period (100-1000) (ms)
DT0023	500	100	1000	Spindle override indicator flickering period (100-1000) (ms)
DT0024	400	100	2000	Feed override knob debounce time (ms)
DT0025	400	100	2000	Spindle override knob debounce time (ms); valid when the machine tool panel is MPU02B
DT0032	10000	0	60000	Liuxin 8-Position Hydraulic Tool Change alarm time (ms)
DT0034	10000	0	60000	AD31 Series Tool Post allowable continuous time upper limit (ms)
DT0035	1000	0	4000	AK31 Series Tool Post lock proximity switch signal detection time upper limit (ms)

2.3.3 Parameter DC

DC address	PLC initial value	Minimum input value	Maximum input value	Meaning
DC0000	50	0	200	The output voltage value of inverter during spindle jog (0.01V)
DC0001	5	0	50	The output voltage value of inverter during spindle automatic gear change (0.01V)

2.3.4 Parameter D

D address	PLC initial value	Minimum input value	Maximum input value	Meaning
D0	4	1	16	Number of tools on a tool post
D1	1	0	5	Internal controlled axis number corresponding to X axis manual movement key (the key is invalid when it is set to 0)

D2	0	0	5	Internal controlled axis number corresponding to Y axis manual movement key (the key is invalid when it is set to 0)
D3	2	0	5	Internal controlled axis number corresponding to Z axis manual movement key (the key is invalid when it is set to 0)
D4	0	0	5	Internal controlled axis number corresponding to the 4th axis manual movement key (the key is invalid when it is set to 0)
D5	0	0	5	Internal controlled axis number corresponding to C axis manual movement key (the key is invalid when it is set to 0)

2.4 PLC(Address A) Alarms (the Followings are Referred to V2.03b)

Standard PLC Alarm (Address A) Instruction		
Address	**No.**	**Message**
A0000.0	1000	Tool change time is too long.
A0000.1	1001	Inversed time is over. The current tool position is inconsistent with the expected one.
A0000.2	1002	Tool change uncompleted
A0000.3	1003	The tailstock function is disabled. M10/M11 command cannot be executed.
A0000.4	1004	Retracting from the tailstock is not allowed during spindle rotation.
A0000.5	1005	The spindle enabling function is closed. Spindle cannot be started.
A0000.6	1006	Protection door is not closed. Machining or spindle start is forbidden.
A0000.7	1007	Low pressure alarm
A0001.0	1008	The chuck cannot be released during spindle rotation
A0001.1	1009	The chuck is not clamping tightly, spindle cannot be started.
A0001.2	1010	Chuck clamping signal is not found during spindle rotation.
A0001.3	1011	The chuck is unclamped. Spindle start is forbidden.
A0001.4	1012	The chuck function is disabled. Command M12/M13 cannot be executed.
A0001.5	1013	Tool post locked signal is not found at the end of tool change.

A0001.6	1014	M code undefined
A0001.7	1015	Undefined alarm
A0002.0	1016	M03, M04 specification error
A0002.1	1017	Automatic gear changing is forbidden during spindle rotation.
A0002.2	1018	D0 setting error (D0 should be less than or equal to 8 and greater than 0)
A0002.3	1019	Undefined alarm
A0002.4	1020	Automatic gear changing is disabled. Check parameter K13.7
A0002.5	1021	Cycle start is not allowed at feeding hold position
A0002.7	1023	Short circuit detected on the machine panel
A0003.0	1024	The specified tool number is larger than the maximum number of tools (D0)
A0003.1	1025	Specified M code invalid
A0003.2	1026	Spindle orientation time is too long
A0003.3	1027	Chuck clamp/release in-position signal is not found
A0004.0	1032	Pre-indexing proximity switch signal is not received
A0004.1	1033	Lock proximity switch signal is not received
A0004.2	1034	The current tool number is inconsistent with the expected one when tool change is finished.
A0004.3	1035	No lock proximity signal when the tool change is finished.
A0004.4	1036	Tool post overheat
A0004.5	1037	D0 setting error (only 8, 10 and 12 are allowed)
A0005.0	1040	Expected tool number not found alarm
A0005.1	1041	Tool post rotation stop and lock signal not found
A0005.2	1042	No lock signal when tool change is finished.
A0005.3	1043	The current tool number is inconsistent with the expected one when tool change is finished.
A0005.4	1044	D0 setting error (only 8 is allowed)

Note: PLC alarm described in the user manual is for the standard ladder, and the concrete PLC alarm messages are referred to the corresponding ladder notes.

Appendix 3 Installation

3.1 GSK988T Appearance Dimension

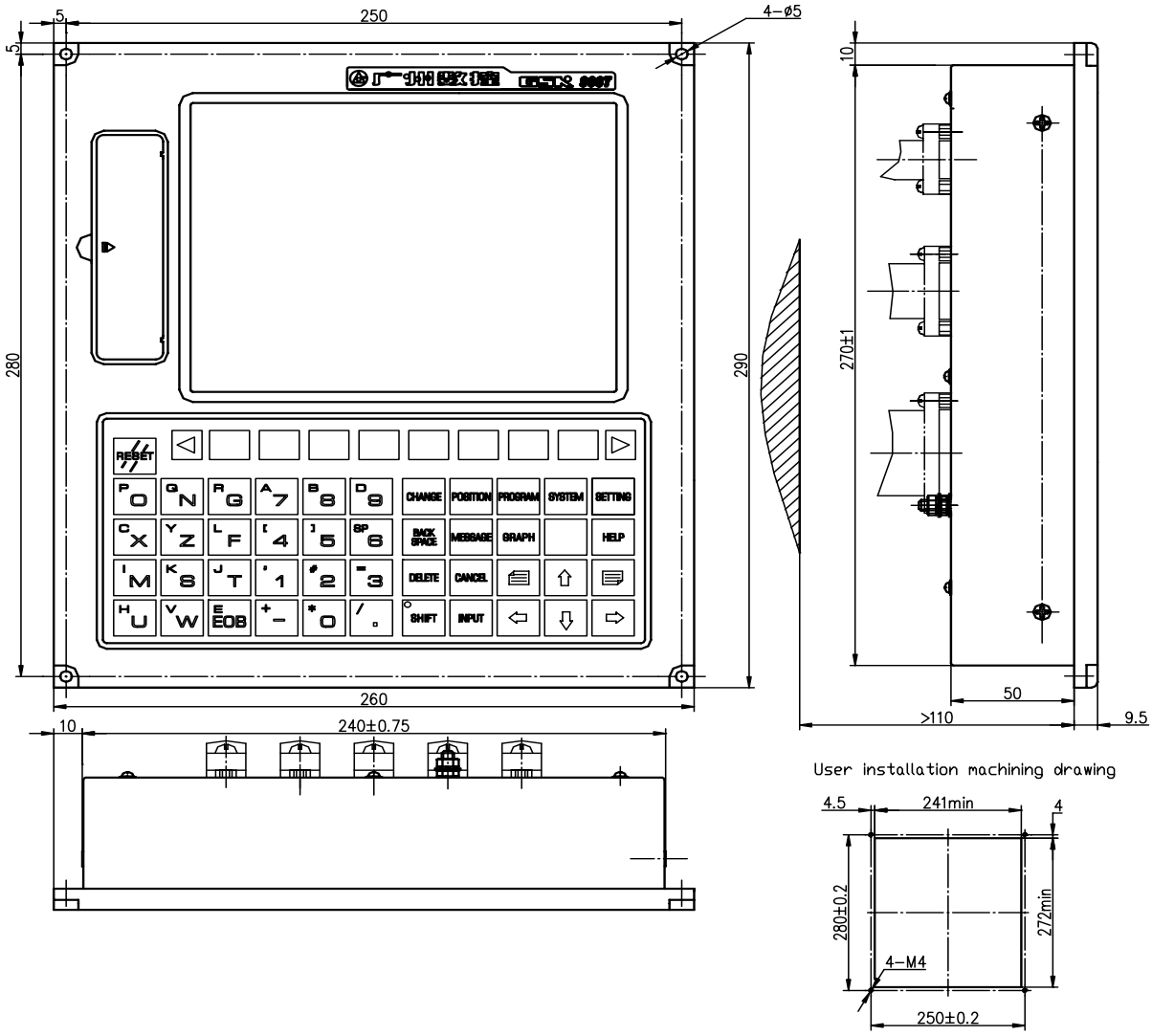


Fig. 3-1 GSK988T appearance dimension

3.2 Machine Operation Panel MPU02A of GSK988T

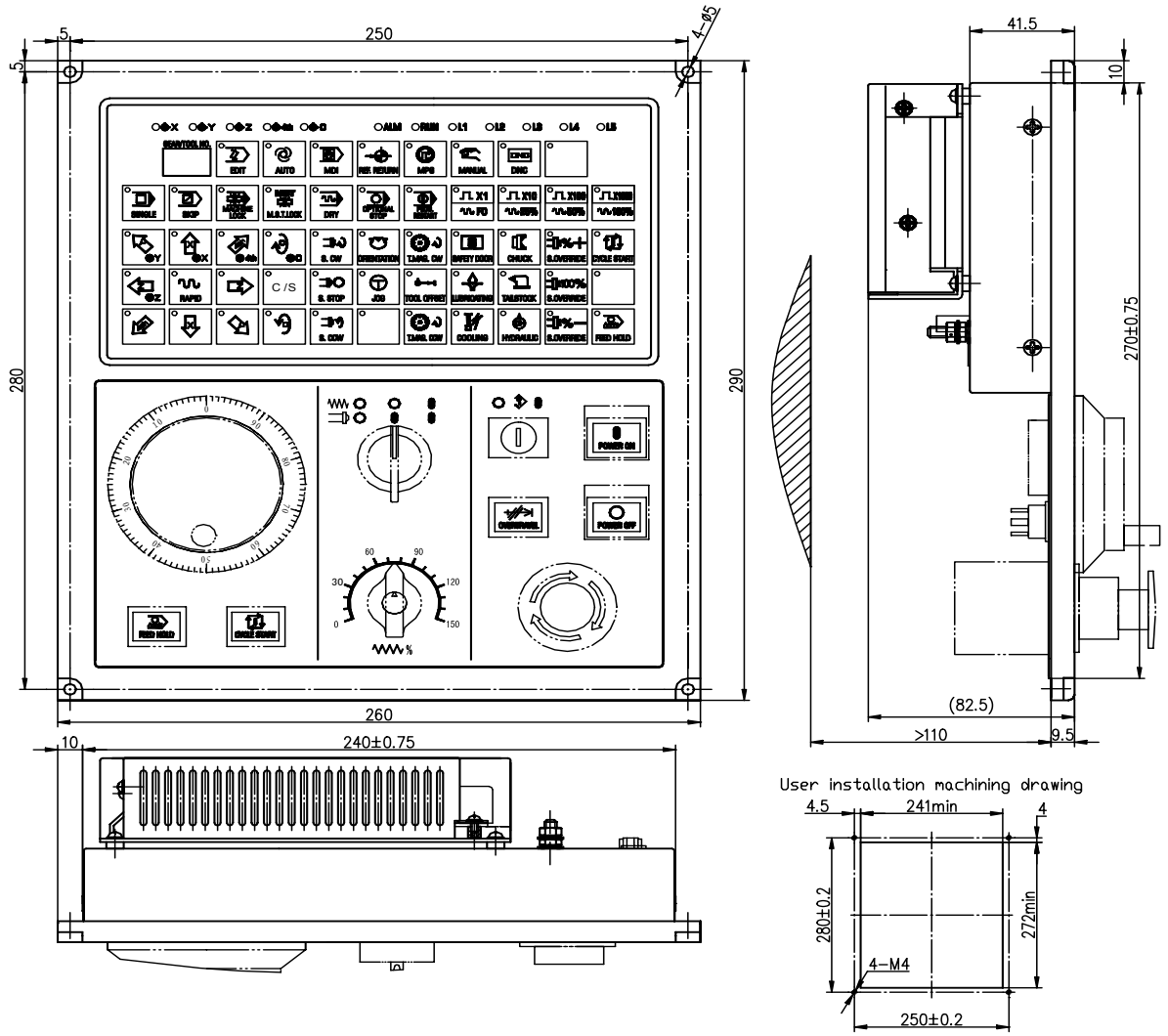


Fig. 3-2 Machine operation panel MPU02A appearance dimension

3.3 Machine Operation Panel MPU02B Appearance dimension of GSK988T

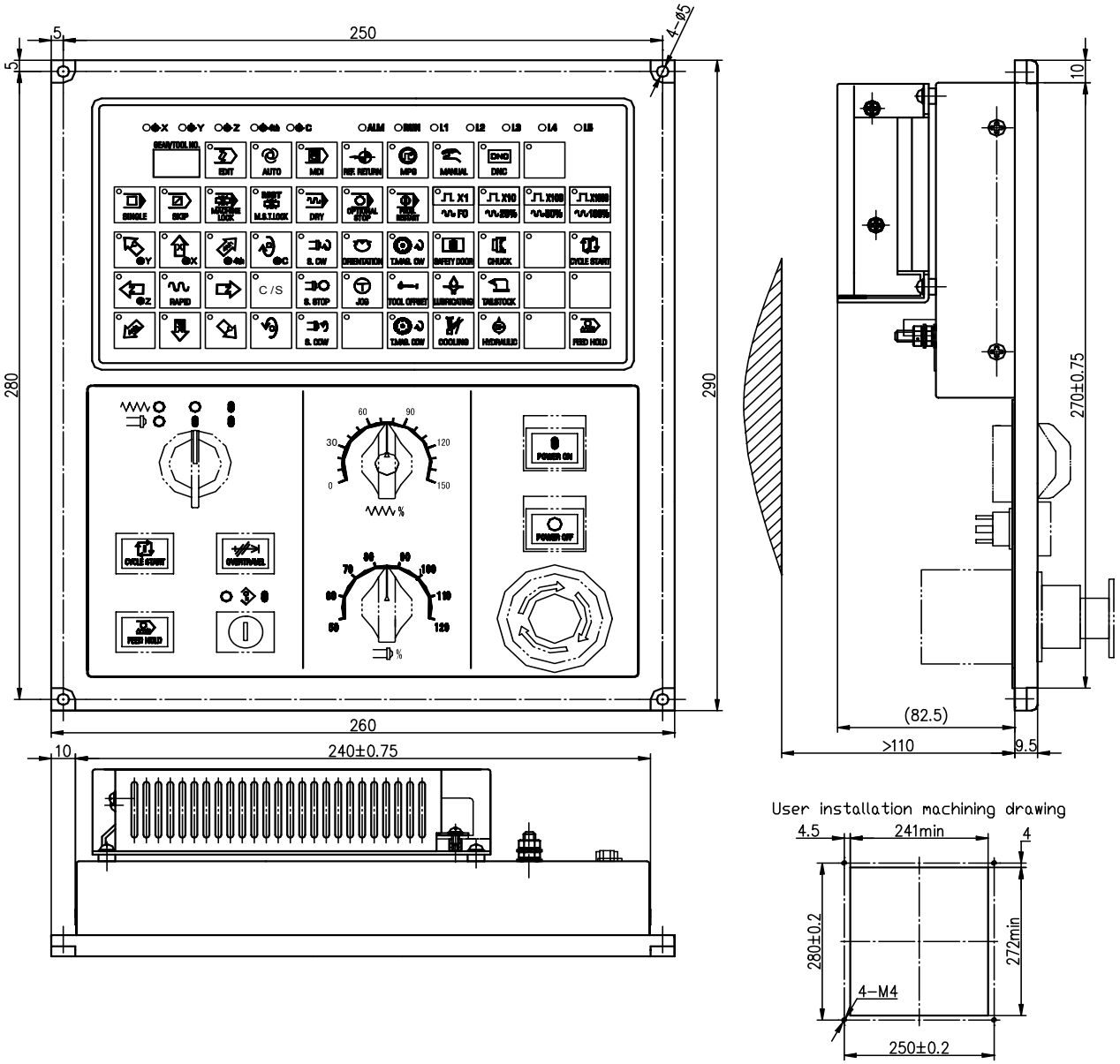


Fig. 3-3 Machine operation panel MPU02B appearance dimension

Appendix

3.4 GSK988T-H Appearance Dimension

Note：GSK988T-H is the horizontal GSK988T CNC System.

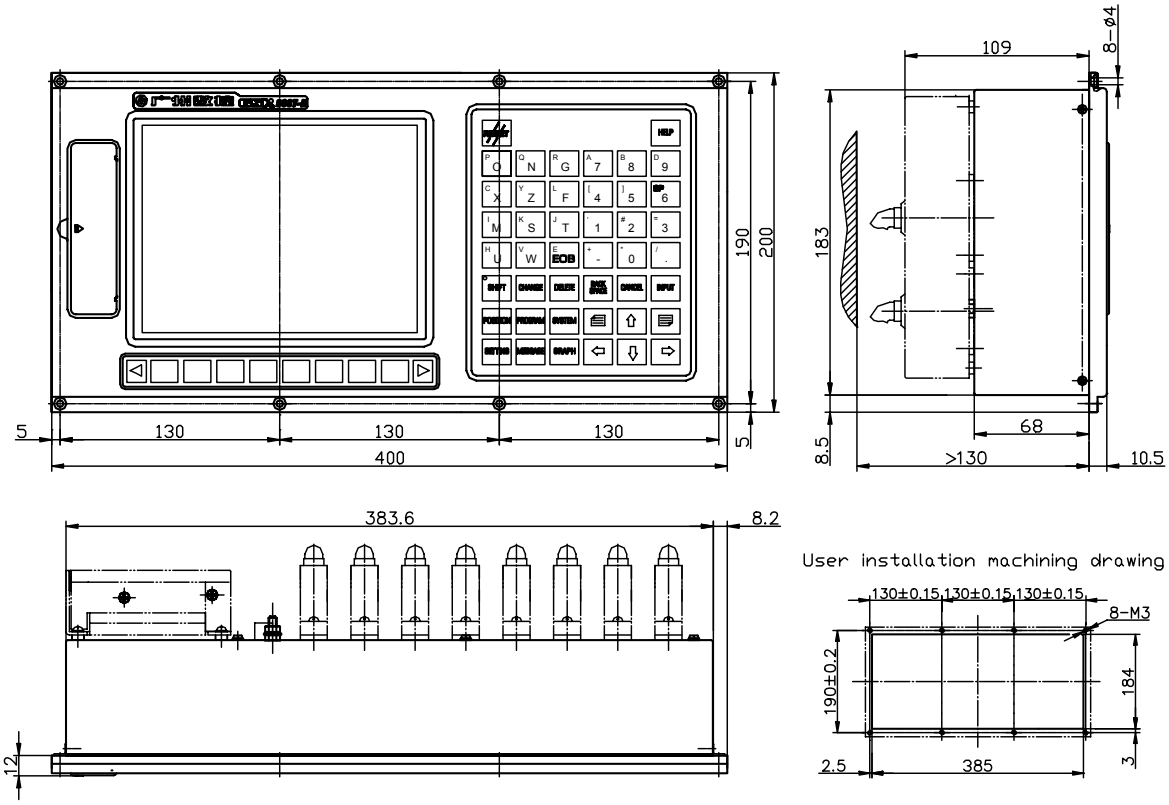
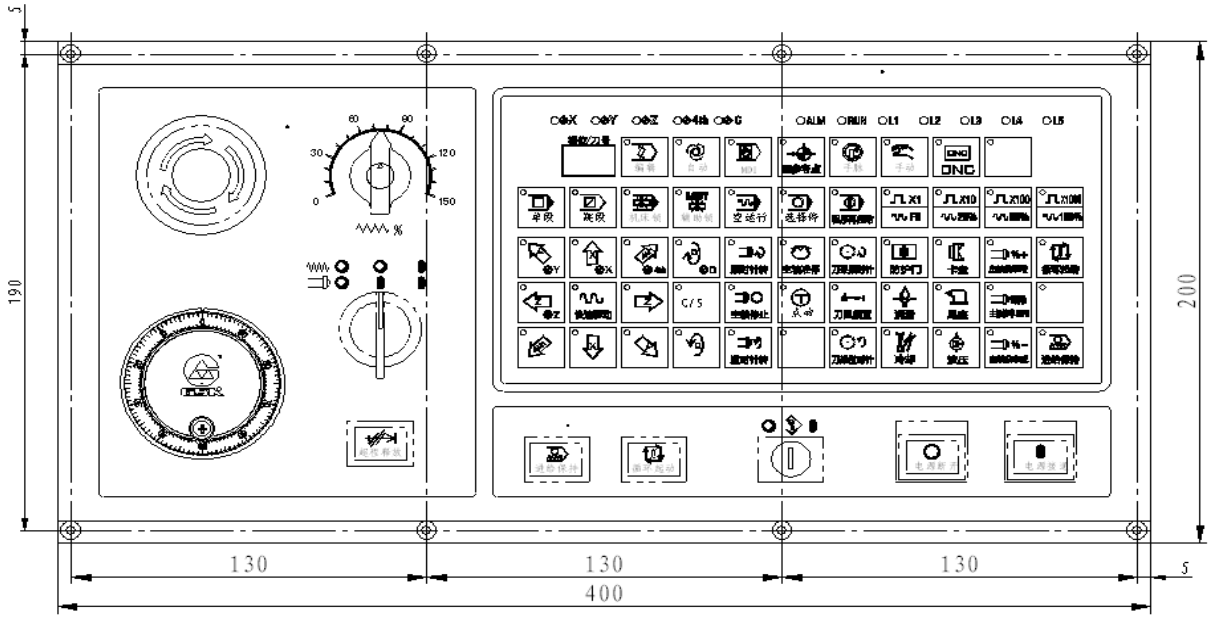


Fig. 3-4 GSK988T—H appearance dimension

3.5 Appearance Dimension of GSK988T-H Operation panel



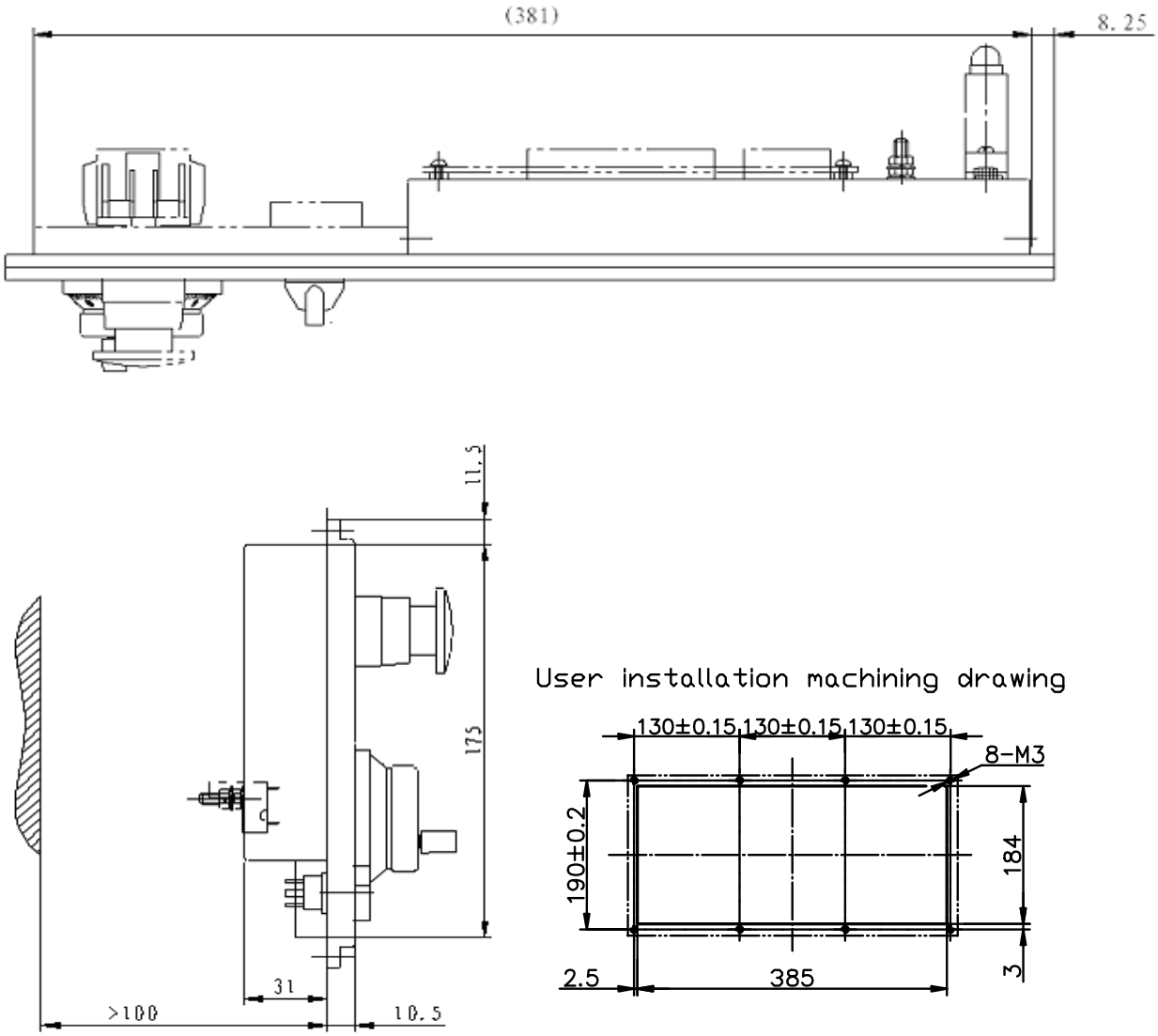


Fig. 3-5 Horizontal operation panel appearance dimension

Appendix 4 Operation List

Type	Function	Operation	Operat-ion mode	Display window	Password level	Program witch	Param-eter switch	Remark
Clear	X incremental coordinate clear	SET REL ,numerical value 0, DATA INPUT		Position				Chapter 3.3.5
	Z incremental coordinate clear	SET REL , ⇩ , numerical value 0, DATA INPUT		Position				
	X tool offset clear	CLEAR		Tool offset setting	2-level, 3-level, 4-level			Chapter 7.1.5
	Z tool offset clear	CLEAR		Tool offset setting	2-level, 3-level, 4-level			Chapter 7.1.5
	Workpiece amount clear	CLEAR PART CNT		Position	2-level, 3-level, 4-level			Chapter 3.1.7
Data setting	Word parameter	DATA INPUT ,Parameter value , DATA INPUT	MDI mode	Parameter	2-level, 3-level,		ON	Chapter 11
	Bit parameter	DATA INPUT ,Parameter value, DATA INPUT	MDI mode	Parameter	2-level, 3-level		ON	
	Macro variable	DATA INPUT , macro variable value, DATA INPUT		Macro variable	2-level, 3-level,			Chapter 3.4.3
	X tool offset incremental input	MEASURE ,X measured value, OK		Tool offset	2-level, 3-level, 4-level			Chapter 7.1.2
	Z tool offset incremental input	MEASURE , Z measured value, OK		Tool offset	2-level, 3-level, 4-level			Chapter 7.1.2
	Tool wear value input	+ INPUT , wear value, OK		Tool offset	2-level, 3-level, 4-level			Chapter 7.1.3

Type	Function	Operation	Operation mode	Display window	Password level	Program witch	Parameter switch	Remark
Search	Search line	LOCATE, line No., OK	Edit mode	Program content	2-level, 3-level, 4-level			Chapter 4.3.1
	Search from home	SEARCH ,Character, FROM TOP			2-level, 3-level, 4-level			
	Search downward from the cursor's current position	SEARCH ,Character, PREV			2-level, 3-level, 4-level			
	Search downward from the current program	SEARCH ,Character, NEXT			2-level, 3-level, 4-level			
	Search the specified program	SERACH ,program name, OK	Edit mode and Auto mode	Program catalog	2-level, 3-level, 4-level			Chapter 4.1.1
	Search system parameters, servo parameters or pitch compensation parameters	SERACH ,Program No., OK		System window				Chapter 3.3
Delete	Character deletion at the cursor	DELETE	Edit mode	Program content	2-level, 3-level	ON		Chapter 4.3.2
	character deletion before the cursor	BACK SPACE	Edit mode	Program content	2-level, 3-level	ON		
	Delete a single block	DEL BLK	Edit mode	Program content	2-level, 3-level	ON		Chapter 4.3.1
	Delete blocks	Select blocks, DELETE	Edit mode	Program content	2-level, 3-level	ON		Chapter 4.3.3
	Delete a program	Search the program to delete, DELETE	Edit mode	Program catalog	2-level, 3-level	ON		Chapter 4.2.3

Type	Function	Operation	Operat -ion mode	Display window	Password level	Program witch	Param -eter switch	Remark
Shortcut key	The cursor moves to the home of a file	CHANGE + ⇧	Edit mode	Program content	2-level, 3-level	ON		Chapter 4.3.3
	The cursor moves to the end of a file	CHANGE + ⇩	Edit mode	Program content	2-level, 3-level	ON		Chapter 4.3.3
	The cursor moves to the home	CHANGE + ⇦	Edit mode	Program content	2-level, 3-level	ON		Chapter 4.3.3
	The cursor moves to the end	CHANGE + ⇨	Edit mode	Program content	2-level, 3-level	ON		Chapter 4.3.3
	Optional block selection	SHIFT + ⇧, ⇩, ⇦, ⇨	Edit mode	Program content	2-level, 3-level	ON		Chapter 4.3.3
	Optional block copy	CHANGE + P O	Edit mode	Program content	2-level, 3-level	ON		Chapter 4.3.3
	Optional block cut	CHANGE + C X	Edit mode	Program content	2-level, 3-level	ON		Chapter 4.3.3
	Optional block past	CHANGE + Q N	Edit mode	Program content	2-level, 3-level	ON		Chapter 4.3.3
Create	Create a program	SERACH ,program name, OK	Edit mode and Auto mode	Program content	2-level, 3-level	ON		Chapter 4.1.2
Rename	Rename a program	SAVE AS ,program name, OK	Edit mode	Program catalog	2-level, 3-level	ON		Chapter 4.2.1
Save as	Save a program as	RENAME ,program name, OK	Edit mode	Program catalog	2-level, 3-level	ON		Chapter 4.2.2
Execution	Execute a program	Select a program, LOAD	Edit mode and Auto mode	Program catalog	2-level, 3-level	ON		Chapter 4.2.2

Type	Function	Operation	Operat-ion mode	Display window	Password level	Program witch	Param-eter switch	Remark
ON/OFF setting	Program ON,OFF	ON：⇦ OFF：⇨	MDI mode	CNC setting	2-level, 3-level			Chapter 3.4.2.1
	Parameter ON, OFF	ON：⇦ OFF：⇨	MDI mode	CNC setting	2-level, 3-level			
	Automatic sequence No ON, OFF	ON：⇦ OFF：⇨	MDI mode	CNC setting	2-level, 3-level			
	Input unit	Metric：⇦ Inch：⇨	MDI mode	CNC setting	2-level, 3-level			

Note 1: "," in "Operation" indicates that the two operations are successive, "+" indicates that the two operations are executed at the same time.

Example: DATA INPUT ,Parameter value, DATA INPUT : press firstly DATA INPUT , and input the parameter value，and then press DATA INPUT again；

CHANGE + Q N : press them simultaneously.

Note 2: The blanks in Operation Mode, Display Window, Password Level, Program Switch and Parameter Switch column indicate that the corresponding switches are not related to their items correspondingly.